

UNCLASSIFIED

AD NUMBER

ADB100945

NEW LIMITATION CHANGE

TO

**Approved for public release, distribution
unlimited**

FROM

**Distribution authorized to DoD only;
Software Documentation; JUL 1985. Other
requests shall be referred to Air Force
Wright Aeronautical Labs.,
Wright-Patterson AFB, OH 45433.**

AUTHORITY

AFWAL ltr, 13 Jul 1987

THIS PAGE IS UNCLASSIFIED

AD B100 945

AUTHORITY:

AFNRL 14 13 July 87

AFWAL-TR-85-2103

Volume II

LABYRINTH SEAL ANALYSIS

Volume II - User's Manual for the Navier-Stokes
Analysis for Labyrinth Seals

January 1986

Final Report for June 1980 - March 1985



R. C. Buggeln
 H. McDonald
 Scientific Research Associates, Inc.
 P.O. Box 498
 Glastonbury, CT 06033

AD-B100 945

Distribution authorized to DOD components only; Software documentation
 July 1985. Other requests for this document shall be referred to
 AFWAL/POTX, W-PAFB, OH 45433 and shall include the statement of terms and
 conditions contained in Atch 21 to AFK 300-6.

WARNING - This document contains technical
 data whose export is restricted by the Arms
 Export Control Act (Title 22, U.S.C.,
 Sec 2751 et seq.) or Executive Order 12470.
 Violation of these export laws is subject
 to severe criminal penalties.

DTIC FILE COPY

DESTRUCTION NOTICE - Destroy by any
 method that will prevent disclosure of
 contents or reconstruction of the
 document.

DTIC
 ELECTE
 APR 16 1986
 S D
 B

AERO PROPULSION LABORATORY
 AIR FORCE WRIGHT AERONAUTICAL LABORATORIES
 AIR FORCE SYSTEMS COMMAND
 WRIGHT PATTERSON AIR FORCE BASE, OH 45433

NOTICE

When Government drawings, specifications, or other data are used for any purpose other than in connection with a definitely related Government procurement operation, the United States Government thereby incurs no responsibility nor any obligation whatsoever; and the fact that the government may have formulated, furnished, or in any way supplied the said drawings, specifications, or other data, is not to be regarded by implication or otherwise as in any manner licensing the holder or any other person or corporation, or conveying any rights or permission to manufacture, use, or sell any patented invention that may in any way be related thereto.

This technical report has been reviewed and is approved for publication.

Gary F. Willmes

GARY F. WILLMES, 2 Lt, USAF
Compressor Research Group

Walker H. Mitchell

WALKER H. MITCHELL
Chief, Technology Branch

FOR THE COMMANDER

H. I. Bush

H. I. BUSH
Director
Turbine Engine Division

If your address has changed, if you wish to be removed from our mailing list, or if the addressee is no longer employed by your organization please notify AFWAL/POTX, Wright-Patterson AFB OH 45433- 6563 to help maintain a current mailing list.

Copies of this report should not be returned unless return is required by security considerations, contractual obligations, or notice on a specific document.

The following notice applies to any unclassified (including originally classified and now declassified) technical reports released to "qualified U.S. contractors" under the provisions of DoD Directive 5230.25, Withholding of Unclassified Technical Data From Public Disclosure.

NOTICE TO ACCOMPANY THE DISSEMINATION OF EXPORT-CONTROLLED TECHNICAL DATA

1. Export of information contained herein, which includes, in some circumstances, release to foreign nationals within the United States, without first obtaining approval or license from the Department of State for items controlled by the International Traffic in Arms Regulations (ITAR), or the Department of Commerce for items controlled by the Export Administration Regulations (EAR), may constitute a violation of law.
2. Under 22 U.S.C. 2778 the penalty for unlawful export of items or information controlled under the ITAR is up to two years imprisonment, or a fine of \$100,000, or both. Under 50 U.S.C., Appendix 2410, the penalty for unlawful export of items or information controlled under the EAR is a fine of up to \$1,000,000, or five times the value of the exports, whichever is greater; or for an individual, imprisonment of up to 10 years, or a fine of up to \$250,000, or both.
3. In accordance with your certification that establishes you as a "qualified U.S. Contractor", unauthorized dissemination of this information is prohibited and may result in disqualification as a qualified U.S. contractor, and may be considered in determining your eligibility for future contracts with the Department of Defense.
4. The U.S. Government assumes no liability for direct patent infringement, or contributory patent infringement or misuse of technical data.
5. The U.S. Government does not warrant the adequacy, accuracy, currency, or completeness of the technical data.
6. The U.S. Government assumes no liability for loss, damage, or injury resulting from manufacture or use for any purpose of any product, article, system, or material involving reliance upon any or all technical data furnished in response to the request for technical data.
7. If the technical data furnished by the Government will be used for commercial manufacturing or other profit potential, a license for such use may be necessary. Any payments made in support of the request for data do not include or involve any license rights.
8. A copy of this notice shall be provided with any partial or complete reproduction of these data that are provided to qualified U.S. contractors.

DESTRUCTION NOTICE

For classified documents, follow the procedures in DoD 5200.22-M, Industrial Security Manual, Section II-19 or DoD 5200.1-R, Information Security Program Regulation, Chapter IX. For unclassified, limited documents, destroy by any method that will prevent disclosure of contents or reconstruction of the document.

AD-B100945L

REPORT DOCUMENTATION PAGE

1a. REPORT SECURITY CLASSIFICATION Unclassified		1b. RESTRICTIVE MARKINGS	
2a. SECURITY CLASSIFICATION AUTHORITY N/A		2. DISTRIBUTION/AVAILABILITY OF REPORT See back	
2b. DECLASSIFICATION/DOWNGRADING SCHEDULE N/A		3. MONITORING ORGANIZATION REPORT NUMBER(S) AFWAL-TR-85-2103, vol, II	
4. PERFORMING ORGANIZATION REPORT NUMBER(S) 310001F		5. MONITORING ORGANIZATION REPORT NUMBER(S) AFWAL-TR-85-2103, vol, II	
6a. NAME OF PERFORMING ORGANIZATION Scientific Research Associates, Inc.	6b. OFFICE SYMBOL (If applicable)	7a. NAME OF MONITORING ORGANIZATION Aero Propulsion Laboratory (AFWAL/POTX) Air Force Wright Aeronautical Laboratories	
6c. ADDRESS (City, State, and ZIP Code) P.O. Box 498 Glastonbury, CT 06033	7b. ADDRESS (City, State, and ZIP Code) Wright Patterson AFB, OH 45433		
8a. NAME OF FUNDING/SPONSORING ORGANIZATION Aero Propulsion Laboratory Air Force Wright Aeronautical	8b. OFFICE SYMBOL (If applicable) AFWAL/POTX	9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER F33615-80-C-2014	
8c. ADDRESS (City, State, and ZIP Code) Wright Patterson Air Force Base, OH 45433	10. SOURCE OF FUNDING NUMBERS		
	PROGRAM ELEMENT NO 62203F	PROJECT NO 3066	TASK NO 10
	WORK UNIT ACCESSION NO 18		
11. TITLE (Include Security Classification) Labyrinth Seal Analysis Volume II - User's Manual for the Navier-Stokes Analysis for Labyrinth Seals (unclassified)			
12. PERSONAL AUTHOR(S) R.C. Buggeln and H. McDonald			
13a. TYPE OF REPORT Final	13b. TIME COVERED FROM 6/80 TO 3/85	14. DATE OF REPORT (Year, Month, Day) January 1986	15. PAGE COUNT 76
16. SUPPLEMENTARY NOTATION			
17. COSATI CODES	18. SUBJECT TERMS (Continue on reverse if necessary and identify by block number) Labyrinth Seals, Navier-Stokes Equations; Computational Fluid Mechanics; Alternating Direction Implicit		
FIELD 21 20	GROUP 05 04		
19. ABSTRACT (Continue on reverse if necessary and identify by block number) This document is a User's Manual for running a computer code for calculating the flow in labyrinth seals. The computer code has the ability to solve the Navier-Stokes equations for two-dimensional and axisymmetric flows for both orthogonal and non-orthogonal coordinate systems. A description of the code, the strategy for running the code, a detailed description of the input, sample input and sample output are presented. A description of a comparison plot program is also presented.			
<p><i>Computational fluid mechanics, Labyrinth Seals; Gas turbine engine applications, Fortran, MINT computer program 12-5 Sept 85</i></p>			
20. DISTRIBUTION/AVAILABILITY OF ABSTRACT <input type="checkbox"/> UNCLASSIFIED/UNLIMITED <input checked="" type="checkbox"/> SAME AS RPT <input type="checkbox"/> EDITION USERS		21. ABSTRACT SECURITY CLASSIFICATION Unclassified	
22a. NAME OF RESPONSIBLE INDIVIDUAL Topham, Keith C., IIT, USAF		22b. TELEPHONE (Include Area Code) (513) 255-6120	22c. OFFICE SYMBOL AFWAL/POTX

Block 3

Distribution authorized to DOD components only; Software documentation July 1985. Other requests for this document shall be referred to AFWAL/POTX, W-PAFB, OH 45433 and shall include the statement of terms and conditions contained in Atch 21 to AFR 300-6.

FOREWORD

This User's Manual describes the computer code which was developed to solve a Navier-Stokes model of the leakage flow through labyrinth seals. The work was accomplished by Scientific Research Associates, Inc., of Glastonbury, Connecticut, under Contract AF33615-80-C-2014 for Labyrinth Seal Analysis. This contract with Allison Gas Turbine Division of General Motors Corporation was sponsored by the Air Force Wright Aeronautical Laboratories, Aeropropulsion Laboratory, United States Air Force, Wright-Patterson AFB, Ohio, with Mr. Charles W. Elrod (AWAFL/POTX) as Project Engineer. Technical coordination was provided by 1st Lt. Keith C. Topham.

This report was submitted in four volumes in May 1985. Volume I summarizes the development of the labyrinth seal Analysis Model. Volume II, this publication, describes the computer code for the Analysis Model. Volume III contains the experimental results, and summarizes the Design Model based on these empirical data. Volume IV presents the User's Manual describing the computer code for the Design Model.

Publication of this report does not constitute Air Force approval of the findings or conclusions presented. It is published only for the exchange and stimulation of ideas.

DTIC
ELECTE
S APR 16 1986 D
B

Accession For	
NTIS GENR	<input checked="" type="checkbox"/>
DTIC TAB	<input checked="" type="checkbox"/>
Unannounced	<input checked="" type="checkbox"/>
Justification	
By _____	
Distribution/	
Availability, Cities	
Avail and/or	
Dist	Special
E-451	



TABLE OF CONTENTS

	Page
1.0 INTRODUCTION	1
2.0 FLOW DIAGRAM	2
3.0 LIST OF MAJOR FORTRAN VARIABLES	7
4.0 MINT SUBROUTINES	11
5.0 LOGICAL FILE UNITS	15
6.0 STRATEGY FOR RUNNING THE MINT COMPUTER CODE	16
7.0 TRANSFORMATION FUNCTION	25
8.0 PROTOCOL FOR GRID POINT AND BOUNDARY IDENTIFICATION	28
9.0 MINT INPUT	30
9.1 \$READ1: Restart Options	31
9.2 \$READ2: Geometric and Grid Options	31
9.3 \$READ3: Flow Conditions	34
9.4 \$READ4: Time Step and Print Control	36
9.5 \$READ5: Boundary Condition Input	39
9.6 \$READ6: Grid Indicator	42
10.0 SAMPLE INPUT	43
11.0 SAMPLE OUTPUT	44
12.0 PLOT PROGRAM USER'S MANUAL	45
12.1 \$PLOT: Control Input	45
12.2 \$PLOT2: Case Options	46
12.3 \$PRFPLT: Profile Plot Input	49
12.4 Sample Input	52
REFERENCES	53
LIST OF SYMBOLS	55
FIGURES	57
TABLES	61

LIST OF FIGURES

	<u>Page</u>
Figure 1 - Typical Coordinate System Construction.	57
Figure 2 - Time Step Selection.	58
Figure 3 - Oh Grid Notation.	59
Figure 4 - A Sample Computational Domain with Embedded Solid Bodies.	60

LIST OF TABLES

	<u>Page</u>
Table 1 - Reference Variables - Units.	61
Table 2 - Sample MINT Input.	62
Table 3 - Sample MINT Output.	63
Table 4 - Sample Plot Input.	76

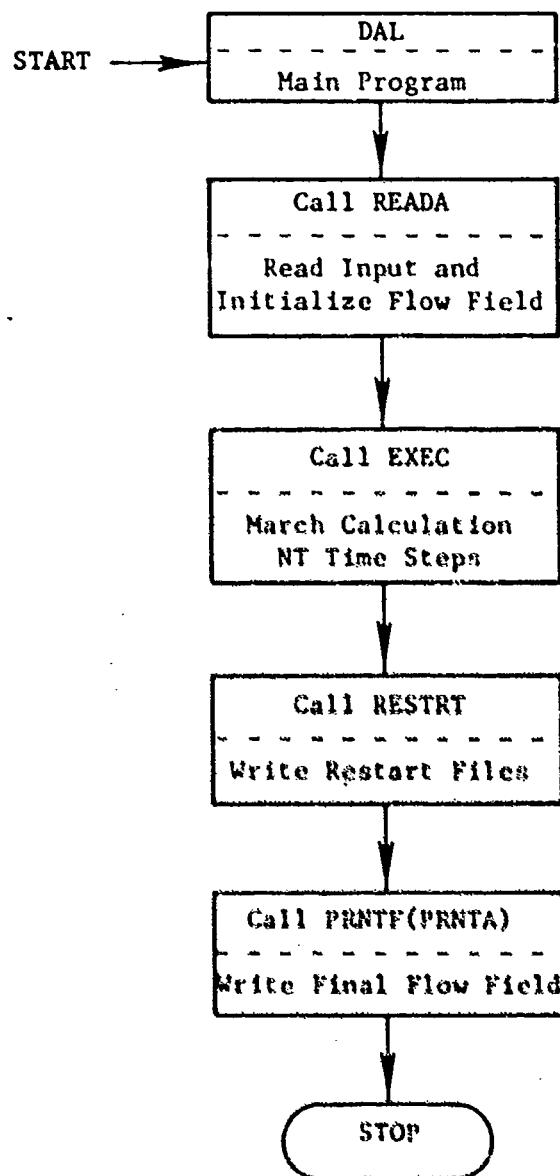
1.0 INTRODUCTION

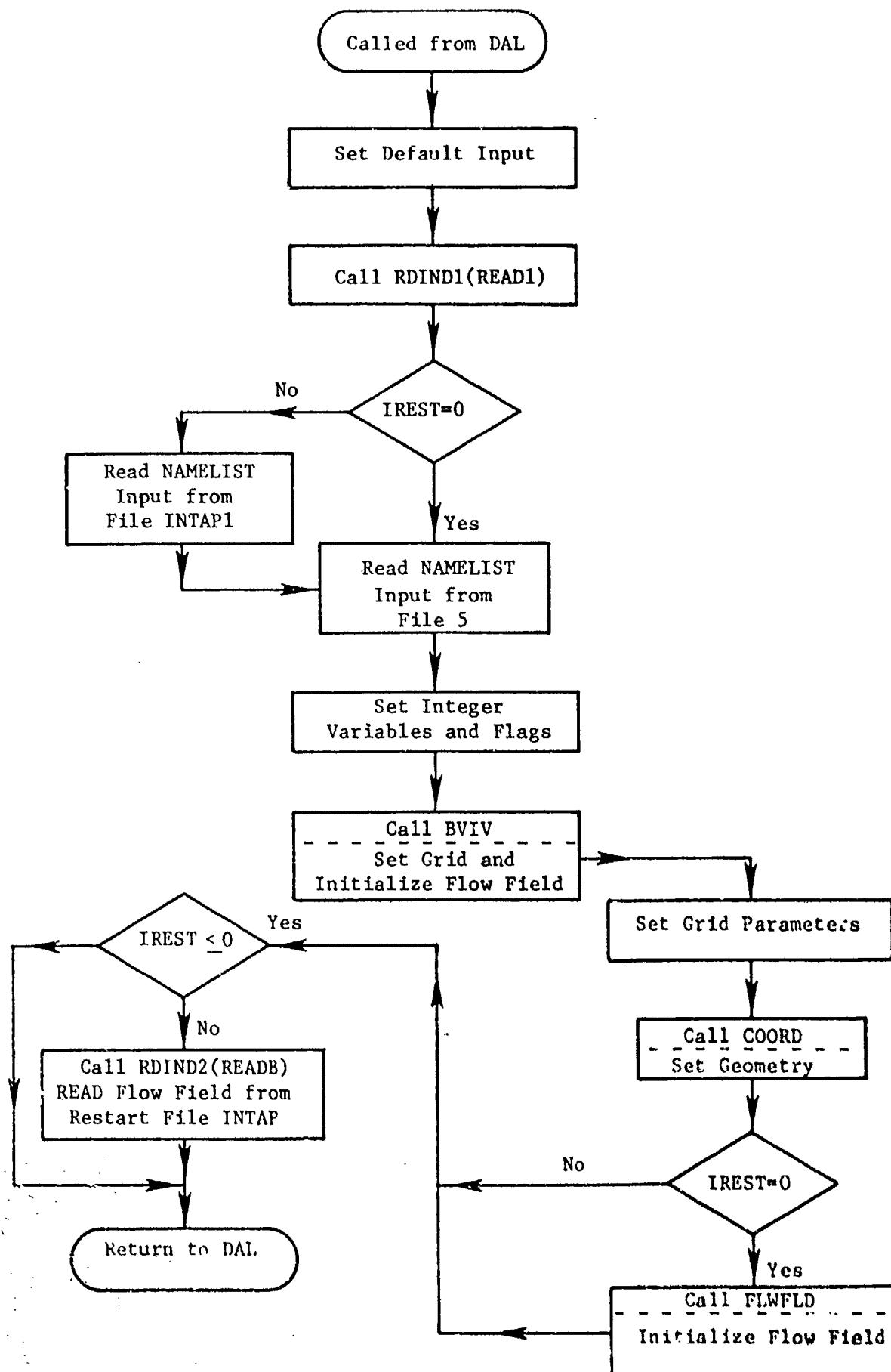
The User's Manual is meant to serve as a guide in helping the user make successful runs with the MINT computer program. The degree of success obtained by the user will depend on the skill of the user and his ability to correctly apply the code to his particular problem. The code will solve the governing equations, subject to the boundary conditions. However, meaningful results will only be obtained if the boundary conditions are appropriate to the problem. In addition, the user must specify viscosity models, initial conditions, a coordinate system and the location of the grid points to adequately resolve the flow. The user with a good knowledge of the physics and a moderate amount of experience should be able to successfully apply the code to a wide variety of problems.

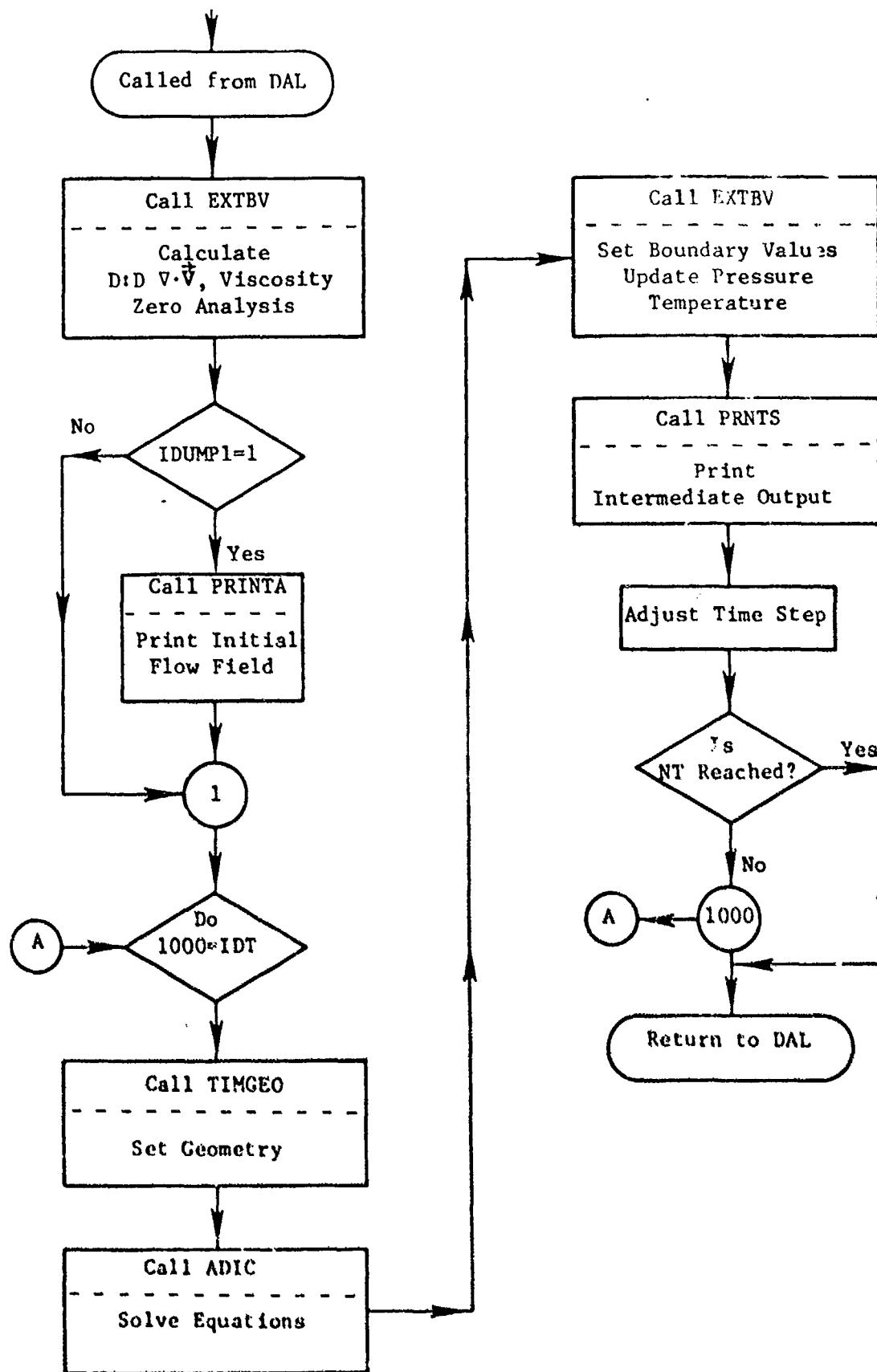
The User's Manual is divided into eleven parts consisting of: (1) a flow diagram of the MINT computer code, (2) a brief description of each subroutine and its use, (3) a list of the major FORTRAN variables and a description of their meaning, (4) a description of the logical file units utilized by the MINT computer code, (5) a brief discourse on the general strategy for the running of the MINT computer code, (6) a description of the method used to distribute grid points on the boundaries, (7) the protocol for grid point and boundary identification, (8) a detailed description of the input required to run labyrinth seal cases, (9) sample input for a labyrinth seal case, (10) sample output for the corresponding case and (11) a description and condensed user's manual for a plot program that can be used with the MINT computer code. For details of the equations solved, boundary conditions utilized and the numerical procedure, the user is referred to Volume I of this report, Ref. 1.

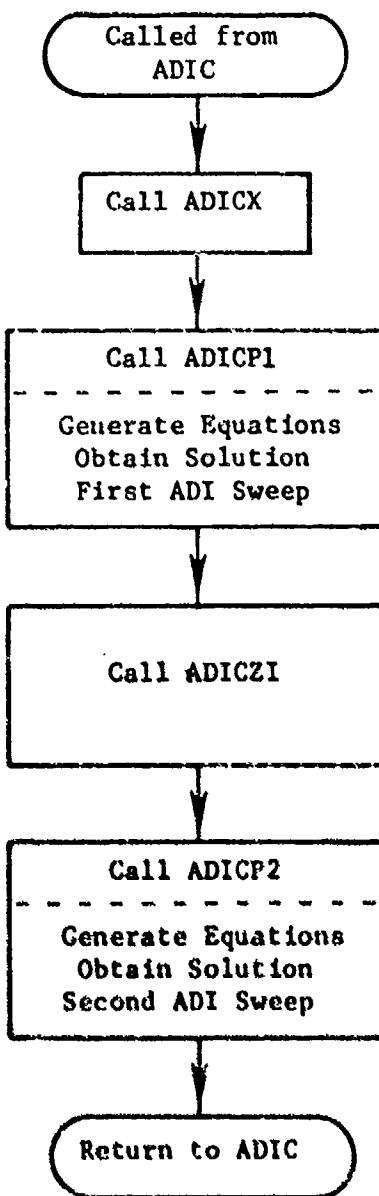
2.0 FLOW DIAGRAM

The purpose of the flow diagram is to help the user understand the basic flow of information within the MINT computer code. Because of the size of the code (approximately 20,000 cards), a detailed flow diagram would be prohibitively large and probably of little value to the user. Therefore, the flow diagram is intended to give the user a general overview of the structure of the code. The interested user is urged to consult the program listing for details. The subroutines contain comment cards to aid the user in the interpretation of the coding.









3.0 LIST OF MAJOR FORTRAN VARIABLES

FORTRAN SYMBOL	COMMON BLOCK	DESCRIPTION
AC(I,J)	BLKI	DEPENDENT VARIABLE ARRAY
ACG(I,J)	BLKI	GEOMETRY DATA ARRAY
AN(I,J)	BLKM	ARRAY STORING TIME TERM LINEARIZED COEFFICIENTS
APR(I,J)	PRNT	PRINT OUTPUT ARRAY
AVANDR	TURB	DAMPING CONSTANT
AVISC(I,J)	MISC2	ARTIFICIAL DISSIPATION PARAMETER
C(I,J,K)	BLKM	COUPLED MATRIX ARRAY STORAGE
CLENG	CREF	REFERENCE LENGTH
CMACH	MISC2	REFERENCE MACH NUMBER
D	VARNO	INDEX FOR DIVERGENCE
D1(I,J,K)	BLKM	ARRAY STORING FIRST SWEEP LINEARIZED COEFFICIENTS
D2(I,J,K)	BLKM	ARRAY STORING SECOND SWEEP LINEARIZED COEFFICIENTS
D3(I,J,K)	BLKM	ARRAY STORING THIRD SWEEP LINEARIZED COEFFICIENTS
DENSR	CREF	REFERENCE DENSITY
DFW(I,J,K)	ADI7	DIFFERENCE WEIGHT ARRAY
DIM1	NOND	INVERSE REYNOLDS NUMBER
DIM2	NOND	REFERENCE PRESSURE/REFERENCE DYNAMIC HEAD
DIM3	NOND	REFERENCE PRESSURE/(REFERENCE DENSITY * REFERENCE ENTHALPY)
DIM4	NOND	1.0/(REY * PRNDL)
DIM12	NOND	2.0 * DIM1
DS	VARNO	INDEX FOR DISSIPATION
DT	MISC2	TIME STEP

FORTRAN SYMBOL	COMMON BLOCK	DESCRIPTION
DTCON	MISC2	INVERSE STEP
DTMAX	MISC2	MAXIMUM ALLOWABLE TIME STEP (SEE \$READY)
DTMIN	MISC2	MINIMUM ALLOWABLE TIME STEP (SEE \$READY)
E(I,J,K)	BLKM	COUPLED MATRIX ARRAY STORAGE
GAMMA	CREF	RATIO OF SPECIFIC HEATS
H	VARNO	INDEX FOR ENTHALPY
I1	MGAUS	LOWER LIMIT FOR MATRIX INVERSION
IADI	ADI1	ADI SWEEP NUMBER
IBC	ADI1	BOUNDARY CONDITION BOUNDARY PARAMETER
IBOUND(I,JDIR)	BCTYP	BOUNDARY TYPE
ICOORD	GEO1	COORDINATE SYSTEM INDEX (SEE \$READ2)
IDT	MISC2	TIME STEP INDEX
IDTADJ	MISC2	TIME STEP CONTROL PARAMETER (SEE \$READ4)
IDUMP1	OUTA	PARAMETER CONTROLLING INITIAL STATION PRINT
IEQ	ADI1	EQUATION NUMBER
IEQBC(I,J,K)	ADI1	BOUNDARY CONDITION INDICATION (SEE \$READ4)
IFSB	PRNT	GRID INDICATOR
IGPRT(I)	GEO1	GEOMETRY PRINT CONTROL (SEE \$READ4)
IL	MGAUS	UPPER LIMIT FOR MATRIX INVERSION
INOUT	CMDISK	IN CORE - OUT OF CORE OPTION
IPRINT	MISC2	PRINT INTERVAL PARAMETER (SEE \$READ4)
IREST	MISC2	RESTART READ CONTROL PARAMETER (SEE \$READ1)
IUNITS	CNEP	DIMENSIONAL UNITS SENTINEL (SEE \$READ3)
IVARPR(I)	MISC2	PRINT PARAMETER (SEE \$READ4)
JADI	ADI1	ADI SWEEP PARAMETER
JG	None	IMPLICIT GRID POINT NUMBER

FORTRAN SYMBOL	COMMON BLOCK	DESCRIPTION
JX	ADI2	DIRECTION-1 GRID POINT INDEX
KZ	ADI2	DIRECTION-3 GRID POINT INDEX
LCORNR	DBLVAL	NUMBER OF RE-ENTRANT CORNERS
LCORNX(I)	DBLVAL	X-LOCATION OF RE-ENTRANT CORNER
LCORNZ(I)	DBLVAL	Z-LOCATION OF RE-ENTRANT CORNER
LX	ADI2	DIRECTION-1 GRID POINT INDEX -
MEQS	ADI1	NUMBER OF EQUATIONS TO BE SOLVED
NT	MISC2	NUMBER OF TIME STEPS TO BE RUN
NTREST	REDUMP	NUMBER OF TIME STEPS BETWEEN RESTARTS (SEE \$READ4)
NTSTEP	MISC2	NUMBER OF TIME STEPS IN A CYCLE (SEE \$READ4)
NUMDX	MISC2	NUMBER OF INTERIOR DIRECTION-1 POINTS (SEE \$READ2)
NUMDZ	MISC2	NUMBER OF INTERIOR DIRECTION-3 POINTS (SEE \$READ2)
NX1	ADI4	FIRST GRID POINT - DIRECTION 1
NX2	ADI4	LAST GRID POINT - DIRECTION 1
NZ1	ADI4	FIRST GRID - DIRECTION 3
NZ2	ADI4	LAST GRID POINT - DIRECTION 3
P	VARNO	INDEX FOR PRESSURE
PCNT1	MISC2	TIME STEP CONTROL PARAMETER
PCNT2	MISC2	TIME STEP CONTROL PARAMETER
PINF	CREF	DIMENSIONAL FREE STREAM PRESSURE (SEE \$READ3)
PREF	CREF	REFERENCE PRESSURE
PRNDL	CREF	PRANDTL NUMBER
PTOT	BCCON	TOTAL PRESSURE
PZERO	BCCON	DIMENSIONAL STAGNATION PRESSURE (SEE \$READ3)
R	VARNO	INDEX FOR DENSITY
REPL	CREF	REYNOLDS NUMBER PER UNIT LENGTH (SEE \$READ3)
REY SAVE	CREP ADI1L	REYNOLDS NUMBER STORAGE FOR * LEVEL DELTA VALUES

FOTRAN SYMBOL	COMMON BLOCK	DESCRIPTION
SN(I)	BLKM	ARRAY STORING SOURCE TERM LINEARIZED COEFFICIENT
SSTEST	MISC2	MAXIMUM CHANGE IN VARIABLE ACROSS TIME STEP
T	VARNO	INDEX FOR TEMPERATURE
TAUW	TURB	WALL SHEAR
TED	VARNO	INDEX FOR TURBULENCE KINETIC ENERGY
TKE	VARNO	INDEX FOR DISSIPATION OF TURBULENCE KINETIC ENERGY
TREF	CREF	REFERENCE TEMPERATURE
TTIME	MISC2	CUMULATIVE TIME
TTOT	BCCON	TOTAL TEMPERATURE
U	VARNO	INDEX FOR DIRECTION-1 VELOCITY
USTAR	TURB	DIMENSIONLESS VELOCITY
V	VARNO	INDEX FOR DIRECTION-2 VELOCITY
VISCL	TURB	LAMINAR REFERENCE VISCOSITY
VISCR	CREF	REFERENCE VISCOSITY
VS1	VARNO	INDEX FOR LAMINAR VISCOSITY
VS2	VARNO	INDEX FOR EFFECTIVE VISCOSITY
W	VARNO	INDEX FOR DIRECTION-3 VELOCITY
WREF	CREF	REFERENCE VELOCITY
XGMAX(I)	GRID1	MAXIMUM COORDINATE VALUE (SEE \$READ2)
XGMIN(I)	GRID1	MINIMUM COORDINATE VALUE (SEE \$READ2)
YPLUS	TURB	DIMENSIONLESS DISTANCE FROM SURFACE

4.0 MINT SUBROUTINES

<u>Subroutine</u>	<u>Purpose</u>
ADIC	Primary control subroutine for ADI procedure.
ADICP1	Control subroutine for first ADI sweep solution - coupled equations.
ADICP2	Control subroutine for last ADI sweep solution - coupled equations.
ADICX	Control subroutine for first ADI sweep.
ADICZI	Control subroutine for last ADI sweep - in core option.
ADICZO	Control subroutine for last ADI sweep - out of core option.
ADIUN1	Control subroutine for first ADI sweep solution - uncoupled equations.
ADIUN2	Control subroutine for last ADI sweep solution - uncoupled equations.
AMATRX	Control subroutine for time and source terms.
ARTVIS1	Calculates artificial dissipation terms.
ATIME	Linearizes time derivatives.
BC	Boundary condition subroutine for general boundary conditions.
BCPM	Specialized boundary conditions subroutine for pressure and momentum.
IVIV	Initiates finite difference grid and geometric parameters.
CONVCT1	Linearizes convection terms for cartesian velocity component formulation.
CONVCT2	Linearizes convection terms for cylindrical polar velocity component formulation.
COORD	Initializes geometry.
CORNER	Sets up logic for re-entrant corners.
CURVT	Linearizes curvature terms.
DAL	Main program.
DELU	Calculates dilatation terms.

<u>Subroutine</u>	<u>Purpose</u>
DIFF1	Linearizes diffusion terms for cartesian velocity component formulation.
DIFF2	Linearizes diffusion terms for cylindrical polar velocity component formulation.
DISFCN	Calculates dissipation function.
DIV	Calculates divergence of the velocity.
DOP	Control subroutine for partial differential equation writing.
DOUBLE	Determines logic for bookkeeping of re-entrant corner logic.
EOS	General equation of state subroutine.
EOSDP	Linearizes pressure gradient terms.
EOSUPI	Updates pressure and temperature; calculates density and enthalpy given pressure and temperature.
EXEC	Master control subroutine for time marching of solution.
EXTBV	Subroutine which updates solution variables from the n^{th} to the $n+1^{st}$ time level and calculates corresponding values for pressure, temperature, viscosity, etc.
FLWFLD	Subroutine to generate initial guess.
GASP	Subroutine which calculates properties of a gas.
GAUSS	Solves the tridiagonal matrix equation $Ax=B$ for x .
GENBC	Control subroutine for boundary conditions - coupled equations.
GENEQ1	Partial differential equation writing subroutine for cartesian velocity component formulation.
GENEQ2	Partial differential equation writing subroutine for cylindrical polar velocity component formulation.
GENUBC	Control subroutine for boundary conditions - uncoupled equations.
GEON	Geometric initialization subroutine.
GSUPS	Calculates nonorthogonal transformation information.
INTGEO	Geometric initialization subroutine.

<u>Subroutine</u>	<u>Purpose</u>
LOADAC	Out of core control subroutine.
MGAUSF	Block tridiagonal matrix solver.
MINMAX	Calculates minimum and maximum extents of a domain.
MINV	Subroutine which solves general matrix equation $Ax=B$ for x .
MIXLEN	Calculates mixing length.
NORMD	Normalization subroutine.
OHGRID	Distributes grid points on a line by the use of error functions - method of Oh.
PRGEO	Controls printout of geometric variables.
PRINT1	Prints out fluid dynamic variables.
PRNTA	Master control subroutine for printout of fluid dynamic variables.
PROB	Calculates probability distribution function.
QUICK	Quick matrix solver.
RDLIST	Reads and writes NAMELIST input data.
READA	Master control subroutine reads in input and initializes flow.
READB	Reads in data and positions units for restart files.
RESTRT	Controls reading and writing of restart files.
RESULT	Control subroutine for printer plots.
ROTATE	Shifts information for out of core option.
SETBV	Control subroutine for performing endcap boundary conditions.
SETDKE	Calculates initial condition for k- ϵ turbulence model.
SORCH1	Calculates for Reynolds number correction terms - cartesian formulation.
SORCH2	Calculates for Reynolds number correction terms - cylindrical polar formulation.
SOURCE	General subroutine which calculates source terms.
SSTST	Calculates maximum charges during a time step.

<u>Subroutine</u>	<u>Purpose</u>
STREAM	Calculates stream function.
TANHYP	Hyperbolic tangent distribution of grid points along a line.
TEMPN	Control subroutine to update temperature and pressure.
TIMGEO	Primary subroutine which calculates geometric transformation information.
TIMLFT	Subroutine which calculates CPU time remaining.
TPLOT	Calculates information needed for printer time plots.
TURBP	Calculates turbulent profile by method of Maise and McDonald.
UPDRUM	Disk control subroutine.
VISCOS	Calculates laminar and turbulent viscosity.
VKPOL	Von Karman - Polhausen velocity profile generator.
WALLPN	Calculates velocity profile information for rough walls.
WHERE	Subroutine which calculates location of nearest walls.
WRDISK	General purpose disk file read, write and position subroutine.
WRPLOT	Subroutine which writes plot file.
WRSLAB	General purpose disk manipulation subroutine.

5.0 LOGICAL FILE UNITS UTILIZED BY THE MINT COMPUTER CODE

The MINT computer code utilizes up to ten (10) logical file units during the execution of a run. In many cases not all ten (10) units are used, and hence in these cases it is not necessary to define all ten (10) units. All references to logical file units in the MINT computer code are accomplished through the use of a FORTRAN name rather than through a specific unit number. Thus if the user desires to change a logical file unit number, this can be accomplished easily. A list of the logical file units utilized by the MINT computer code, their FORTRAN names, the default value unit number and a brief description of the use of the unit is presented below.

<u>FORTRAN Name</u>	<u>Default Unit Number</u>	<u>Description</u>
KTAPE	1	Plot file.
MINP	5	Input file for NAMELIST Input.
MPRT	6	Printed output unit.
IOTAPE	10	Output unit number for dependent variable array.
INTAPE	10	Input unit number for dependent variable array.
NUNIT1	11	Scratch file for out of core option.
NUNIT2	12	Scratch file for out of core option.
IOTAPI	20	Output unit number for NAMELIST restart data.
INTAPI	20	Input unit number for NAMELIST restart data.
NTAG	21	Input unit for geometric data - X,Z pairs written in records of length NDUMX+2.

6.0 STRATEGY FOR RUNNING THE MINT COMPUTER CODE

The following discussion is meant to give the user a basic understanding of the strategy for successfully running the MINT computer code. It is suggested that the user read and understand this section before attempting to prepare input for the MINT code. Basically the strategy for running the MINT code can be divided into seven areas:

- (1) Specification of the system of units to be employed, and reference flow conditions,
- (2) generation of the coordinate system in which the calculation is to be performed,
- (3) specification of the boundary conditions to be used,
- (4) generation of the initial conditions
- (5) selection of the time step to insure rapid convergence of the solution
- (6) solution of the governing equations with a k- ϵ turbulence model and
- (7) solution of the governing equations with rotation.

Preparation of the input for the MINT computer code first requires the choice of a system of units to be used for the nondimensionalization of the governing equations (all variables are nondimensionalized with respect to the free stream conditions). Two options are available in the MINT code, viz., the MKS system and the English system corresponding to values of the FORTRAN variable IUNITS= 1 and 2 respectively. Table 1 shows the various reference variables, and the units in both the MKS and English systems. Reference condition input for the MINT code can be divided into two categories which are referred to as: (1) flight conditions and (2) wind tunnel conditions and refer to the types of flow condition information that would normally be available to the user. Adequate discussion of the two types of reference conditions requires the use of several simple relationships which tie together the various flow conditions. They are the perfect gas law

$$P_{\infty} = \rho_{\infty} R T_{\infty} \quad (1)$$

the definition of Mach number

$$M_{\infty} = \frac{U_{\infty}}{\sqrt{\gamma R T_{\infty}}} \quad (2)$$

the isentropic relationships along a streamline for a perfect gas with a constant specific heat, $c_{p\infty}$,

(3)

$$\frac{P_0}{P_\infty} = \left(1 + \frac{\gamma-1}{2} M_\infty^2\right)^{\gamma/\gamma-1}$$

(4)

$$\frac{T_0}{T_\infty} = 1 + \frac{\gamma-1}{2} M_\infty^2$$

The definition of Reynolds number per unit length

(5)

$$\frac{Re}{l_\infty} = \frac{\rho_\infty U_\infty}{\mu_\infty (T_\infty)}$$

Sutherlands viscosity law, which assumes that the viscosity of a thermally perfect gas is a function of temperature (T_∞) alone,

(6)

$$\mu_\infty (T_\infty) = \frac{c_1 T_\infty^{3/2}}{c_2 + T_\infty}$$

the specific heat relationship for a perfect gas

(7)

$$c_{p\infty} = \frac{\gamma}{\gamma-1} R$$

and the definition of enthalpy for a constant specific heat perfect gas,

(8)

$$h_\infty = c_{p\infty} T_\infty$$

When flight conditions are given, typically a reference length, l_∞ , free stream Mach number, M_∞ , static temperature, T_∞ , and static pressure, P_∞ , are specified. Given these variables the free stream or reference velocity, U_∞ , can be calculated from Eq. (2), while Eq. (1) can be used to calculate the free stream density, ρ_∞ . Eq. (6) yields the free stream viscosity, μ_∞ , and Eq. (5) can be used to calculate the Reynolds number per unit length. Eqs. (3) and (4) yield the free stream stagnation pressure and temperature while Eqs. (7) and (8) yield the free stream specific heat and enthalpy.

When wind tunnel conditions are given, the normal input information usually consists of the reference length, l_∞ , Mach number, M_∞ , the Reynolds number per unit length, Re/l_∞ , and either the static pressure, P_∞ , or the stagnation pressure, P_0 . By combining Eqs. (1), (2), (5) and (6) a quadratic equation for T_∞ can be obtained in terms of known variables, i.e.,

$$T_\infty = \frac{8 + \sqrt{B^2 + 4c_1 c_2}}{2c_1} \quad (9)$$

where

$$B = \frac{\frac{\sqrt{\gamma}}{R} M_\infty P_\infty}{\frac{Re}{l_\infty}} = \frac{\frac{\sqrt{\gamma}}{R} M_\infty P_0}{\frac{Re}{l_\infty} \left(1 + \frac{\gamma-1}{2} M_\infty^2\right)^{\gamma/(\gamma-1)}} \quad (10)$$

Eqs. (1), (2), (4), (6), (7) and (8) can then be used to calculate free stream density, velocity, stagnation temperature, viscosity, specific heat and enthalpy respectively.

Other categories of input are also possible and any of the free stream or reference conditions can be calculated by the appropriate combination of Eqs. (1)-(8). For the case of labyrinth seal calculations, the flow information obtained often consists of a mass flux entering the labyrinth seal inlet channel, the entry temperature, T_∞ , an entry pressure, P_∞ , and the seal geometry. From the seal geometry an arbitrary reference length, l_m , can be chosen (often the clearance gap height or the inlet channel height, i.e., rotor to land distance is used). Given the inlet or free stream temperature, Eq. (6) can be used to calculate the free stream viscosity, μ_∞ . Since the mass flux, $\rho_\infty U_\infty$, is known, Eq. (5) can be used to calculate the Reynolds number per unit length, Re/l_∞ . (In reality the mass flux will have to be multiplied by unity plus a blockage factor, b , to obtain the value of $\rho_\infty U_\infty$. The blockage factor is obtained from a knowledge of the inlet channel flow characteristics). Eq. (1) can then be used to calculate the free stream density, ρ_∞ , given the free stream pressure and temperature. Since the mass flux, $\rho_\infty U_\infty$, is known the free stream velocity U_∞ , can be calculated and hence the free stream Mach number, M_∞ , can be calculated. Finally at this point the flow conditions can be input via the wind tunnel option.

The MINT computer code has a very general coordinate capability, i.e., calculations can be performed in general nonorthogonal coordinates. Both two-dimensional (ICOORD = 0) and axisymmetric (ICOORD = 1) nonorthogonal coordinate systems can be utilized. The former is primarily utilized to simulate laboratory test conditions while the latter is used to simulate actual rotating hardware components. Geometric information can be calculated externally to the MINT computer code, in which case this data is read into the MINT code from unit NTAG = 21. In general, this capability is utilized for complex geometric configurations where it would be inconvenient to code such information into the MINT program. The MINT code has the capability of internally calculating the geometry for a wide variety of seal configurations. Basically, any seal configuration consisting of knives and lands with either wall rotating that is constructed from a series of alternating nominally horizontal and vertical straight lines can be analyzed. Thus, for example, rectangular, slanted and tapered knives in both stepped seal and straight through seals can be analyzed (see Fig. 1).

The process of constructing these 'stick-like' labyrinth seal configurations is as follows: (see Fig. 1).

(1) Base lines (dotted lines of Fig. 1) are drawn on the bottom and top and on the left-hand boundaries;

(2) Grid points are next distributed on the bottom and top boundaries and on the left-hand boundary by use of the transformation technique of Oh (Ref. 2). This technique allows selected grid points to be constrained to a specific physical location and thus the surfaces of the rotor and knives can be specified;

(3) Lines emanating from the grid points on the left-hand side are then drawn horizontally across to the right-hand side of the domain; and

(4) Grid points on the top and bottom boundaries are then connected with straight lines thus completing the coordinate system.

The method of distributing grid points on the boundaries is described in detail in the next section. The basic procedure is to concentrate a relatively large number of grid points in regions of the flow where large gradients of the physical variables are expected. For labyrinth seal configurations this usually means in the vicinity of boundary layers, in the vicinity of the gap between the knives and the land and in the vicinity of shock waves (if they exist). In general, it is desirable to

start the computation domain far upstream of the first knife, i.e. on the order of four or five channel heights upstream of the knife. Since there will often exist a large recirculation zone downstream of the last knife, the downstream region has to include at least the entire extent of this recirculation zone. The procedure normally employed is to use relatively large streamwise grid spacing at the upstream and downstream boundaries, and relatively fine streamwise resolution in the vicinity of the seal assembly.

A wide variety of boundary conditions are available as input to the user of the MINT computer code. In addition, there are default values of boundary conditions which can be activated by identifying the boundaries with the FORTRAN variable IBOUND(KSURF, IDIR) where KSURF is the surface number with respect to the IDIRth direction (for discussion of convention used for KSURF and IDIR see section on Protocol for Grid Point and Boundary Identification). Values of IBOUND = 1,2,3 or 4 correspond to an inlet, a symmetry surface, an exit plane or a wall, respectively. Default values of the boundary conditions (FORTRAN variable IEQBC(KSURF, IDIR, IEQ)) associated with specific values of IBOUND are described in the NAMELIST input description section. Any default value of IEQBC can be overwritten by the inputting of a value of IEQBC in the NAMELIST input.

One boundary condition that needs some explanation is the so-called 'two-layer model' which is used as one of the default boundary conditions on an inlet. This boundary condition assumes that the flow in an inlet is divided into two regions or layers, a central inviscid core where the stagnation pressure is constant, and an attached boundary layer(s) where the pressure is constant and the velocity profile has a prescribed form. The prescribed form (but not the magnitude) of the boundary layer velocity profile must be supplied by the user. The MINT code presently has two methods of specifying the form of the boundary layer profile(s):

(1) laminar profiles are generated in SUBROUTINE VKPOL where a von Karman-Pohlhausen (Ref. 3) profile is generated, and (2) turbulent profiles are generated in SUBROUTINE TURBP where a Maise-McDonald (Ref. 4) profile is generated. One feature of the two-layer model is that the mass flux will be determined from the solution of the governing equations, and not from an imposition of the inlet profiles of velocity and density.

Since the steady state solution is obtained by time marching the governing equations until the solution ceases changing, an initial guess

of all flow variables must be generated. Because of the wide variety of the geometric configurations considered, it would be impossible to develop a single technique for obtaining initial conditions. Rather, the approach taken in this study is to assume that initially the flow is at rest with all conditions equal to the stagnation values. Then the back pressure is lowered over a prescribed number of time steps until the desired downstream pressure is obtained. As the pressure pulse proceeds from the downstream boundary to the inlet, the flow is gradually accelerated or drawn through the device until a steady state solution is obtained. The technique used is to gradually lower the back pressure, P_B , from the initial stagnation value, P_0 , to the desired back pressure, P_a , by use of the relationship

$$P_B = P_0(1-f) + fP_a \quad (11)$$

where

$$f = \frac{1 - \cos \left\{ \pi \max \left[0, \min \left(1, -\frac{n-n_1}{n_2-n_1} \right) \right] \right\}}{2} \quad (12)$$

and n is the present time step, n_1 is the time step when the back pressure, P_B , starts to change from its initial value, P_0 , and n_2 , is the time step when P_B is equal to the final pressure P_a . When $n \leq n_1$, $f = 0$ hence $P_B = P_0$. When $n \geq n_2$, $f = 1$ hence $P_B = P_a$. At intermediate values of n , i.e., $n_1 < n < n_2$, f varies as

$$f = \frac{1 - \cos \left(\pi \frac{n-n_1}{n_2-n_1} \right)}{2} \quad (13)$$

At values of $n = n_1$ and $n = n_2$, $\frac{df}{dn} = 0$.

The choice of the numerical time step size is critical in obtaining an economic converged solution to the governing equations. Since in this study only steady state solutions were considered, there is no need to require that transient accuracy be maintained. One means that is used to rapidly obtain a steady state solution is to utilize a time step that varies with the location within the computational domain. The approach used here is to use a small time step in the region of the knives and larger time steps in the regions both upstream and downstream

of the seal. This has the advantage of allowing the propagation of the pressure wave (caused by the lowering of the back pressure) to rapidly, i.e., in a few time steps, travel upstream. In the region of the knives, where large transverse and streamwise velocities and velocity gradients will occur, stability can be maintained with the use of the smaller time steps. In the region upstream of the knives, a larger time step is again employed as in this region the gradients can be expected to be small.

The spatial time step variation, Δt , is controlled by use of the function

$$\frac{1}{\Delta t} = \frac{1}{\Delta t_B} \left[\frac{1}{f_1(\Delta t_1 - 1) + 1 + f_2(\Delta t_2 - 1)} \right] \quad (14)$$

where

$$f_1 = 1 - \frac{1 - \cos \left\{ \pi \max \left[0, \min \left(1, \frac{z - z_1}{z_2 - z_1} \right) \right] \right\}}{2} \quad (15)$$

$$f_2 = \frac{1 - \cos \left\{ \pi \max \left[0, \min \left(1, \frac{z - z_3}{z_4 - z_3} \right) \right] \right\}}{2} \quad (16)$$

and where z, z_1, z_2, z_3 , and z_4 are shown in Fig. 2. Δt_1 is the ratio of the time step upstream of z_1 to the time step over the knife assembly, Δt_B , and Δt_2 is the ratio of the time step downstream of z_4 to the time step over the knife assembly. The variables f_1 and f_2 serve a similar function to that of the previously described f in the back pressure formulation.

The basic approach is to initially take as large a time step as possible while still maintaining numerical stability. In this manner the basic features of the flow are rapidly developed and then the time step can be reduced to obtain the converged steady state solution. Typically the user must, for a given class of geometric configurations, experiment with the initial time step. Usually the back pressure can be dropped over approximately 25 time steps, and with an additional 25 time steps the basic flow patterns will be developed. The important thing to observe during the initial time steps is the propagation of the pressure wave upstream. In addition, it is desirable to watch the relative change (FORTRAN variable SSTST) of the dependent variables, and the values of the maximum residuals (FORTRAN variable RESMAX). Initially, both SSTST and RESMAX should be

relatively large. However, after the back pressure has been lowered to its final value both of these variables should start to decrease. By the 50th time step the streamwise velocity on the inlet plane should approach within 15%-20% of its final value. The values of SSTST and RESMAX should be continuously monitored during a run as these are the parameters which are used to determine if the solution is converged.

After the basic flow patterns have been developed, the large time step should be decreased to allow a converged solution to be obtained. As the MINT code is presently configured, the time step can only be changed during a restart. Hence, the overall strategy is to make an initial run at a large time step, to catalogue the restart files and then to make subsequent runs at successively smaller time steps until a converged solution is obtained. The above guidelines have been successfully used in this study. However, it is important to stress that a certain amount of experience in time step selection is necessary, and several value judgements must be made in obtaining converged solutions.

Solution of the governing equations with a $k-\epsilon$ turbulence model requires the solution of two additional partial differential equations. The procedure used in this study is to first solve the flow equations of conservation of mass, momentum and energy with a mixing length turbulence model. The flow is then frozen in time and the k and ϵ partial differential equations are solved to obtain a converged k and ϵ field (for the frozen flow). Then the flow equations and the k and ϵ equations are solved simultaneously until a converged solution is obtained. The above is done with the input FORTRAN variables IVISC and ISETKE. Initially IVISC is set equal to 3, the mixing length turbulence model option. A converged solution is obtained, and restart files catalogued. Next on a restart run IVISC is set equal to -5, and ISETKE is set equal to the last restart time step. An initial guess of k and ϵ will then be obtained in SUBROUTINE SETDKE, and the k and ϵ equations will be solved with a frozen flow field. Once a converged $k-\epsilon$ solution is obtained, a restart must be catalogued and a new run initiated with IVISC set equal to 5 at which time the flow equations and the k and ϵ equations will be solved simultaneously. It is possible (but not recommended) that the intermediate step of solving the $k-\epsilon$ equations with frozen flow can be eliminated.

In this case the mixing length solution would be obtained and IVISC would be changed from 3 to 5 and the flow equations and the k and ϵ equations would be solved simultaneously. The value of ISETKE would still have to be set to determine the time step when the transition from mixing length to k - ϵ turbulence model would occur.

Solution of the governing equations with rotation requires that an addition swirl momentum equation be solved. The procedure used in this study was to first solve the governing equations with no rotation and then gradually to increase the rotational speed of the appropriate boundaries from zero to the desired rotational speed. This is done by enforcing a boundary condition on the rotating surfaces. The rotational velocity on the boundaries is again obtained with the use of a cosine function of the form

$$V_\theta = frV_{\theta_f} \quad (17)$$

where r is the radius (at the desired point on the rotating surface), V_{θ_f} is the final rotational speed and

$$f = \frac{1 - \cos \left\{ \pi \max \left[0, \min \left(1, \frac{n - n_1}{n_2 - n_1} \right) \right] \right\}}{2} \quad (18)$$

where in this case n_1 is the first time step where rotation starts and n_2 is the time step when the final rotational velocity, V_{θ_f} , is achieved.

It is possible to save some computational time by not solving the rotational (swirl) momentum equation before time step n_1 by setting ISWIRL = 0, and running until time step n_1 . At this point a restart must be catalogued and a new run initiated with ISWIRL = 1. This procedure is optional.

7.0 TRANSFORMATION FUNCTION

The distribution of grid points on boundaries is accomplished by the use of a transformation technique developed by Oh and described in Ref. 2. If y and η designate the independent variables in physical and computational (grid point) space respectively, the transformation function

$$f = \frac{dy}{d\eta} \quad (19)$$

can be integrated to yield

$$y = \int_{\eta_{\min}}^{\eta} f(\eta) d\eta + y_{\min} \quad (20)$$

A convenient transformation function is composed of a series of N complementary error functions of the form

$$f(\eta) = \frac{dy}{d\eta} = \beta_0 + \frac{1}{2} \sum_{j=1}^N \left\{ \operatorname{erfc} \left[\frac{\gamma}{a_j} (\eta - \eta_{p_j}) \right] - [1 + \operatorname{sign}(a_j)] \right\} \beta_j \quad (21)$$

The j^{th} complementary error function is centered in computational space at location η_{p_j} (which is referred to as a pivot point), and a_j is the width in computational space in which 90% of the grid size variation takes place. At computational space location η_{p_j} , the values of $f(\eta)$ will assume a local maximum. γ is a convenient scaling constant for a_j ($\gamma = 1.163 \times 2 = 2.326$; $\operatorname{erfc}(1.163) \approx 0.10$). In the limit as $a_j \rightarrow 0$, β_j is the j^{th} step height for pivot j , i.e., the difference in grid spacing on either side of η_{p_j} .

Substitution of Eq. (21) into Eq. (20) and integration yields

$$y - y_{\min} = \beta_0(\eta - \eta_{\min}) + \sum_{j=1}^N \frac{1}{2} \left\{ \frac{a_j}{\gamma} \left[\theta_j(\eta) - \theta(\eta_{\min}) \right] - [1 + \operatorname{sign}(a_j)] [\eta - \eta_{\min}] \right\} \beta_j \quad (22)$$

where

$$\begin{aligned}\theta_j(\eta) &= \frac{\gamma}{a_j} (\eta - \eta_{pj}) \operatorname{erfc} \left[\frac{\gamma}{a_j} (\eta - \eta_{pj}) \right] \\ &- \frac{1}{\sqrt{\pi}} e^{-[\gamma/a_j(\eta-\eta_{pj})]^2}\end{aligned}\quad (23)$$

The technique used in this study is to constrain the values of the physical coordinate, y_{ck} , at specific values of the computational coordinate, η_{ck} . At interior points the η_{ck} 's are referred to as interior cluster points. At the two end lines, there exist pairs of computational and physical points η_{\min} , y_{\min} , and η_{\max} , y_{\max} which are referred to as end cluster points. In this formulation (see Fig. 3), two pivot points are associated with each interior cluster point and one pivot point is associated with each end cluster point ($\eta_{p\min}$ and $\eta_{p\max}$). Thus, if there are k interior cluster points, the total number of pivot points is $N = 2k+2$. To determine the functional relationship $y(\eta)$ described by Eq. (22), requires (if the values of η_{pj} and a_j are prescribed) that the values of the $(N+1)$ β 's in that equation be calculated. Both Eq. (21) and Eq. (22) are linear equations with respect to the β 's. By constraining the values of the physical coordinates at the interior and end cluster points, $(k+1)$ linear independent equations in the β 's are obtained. By further constraining the values of the slope, $f(\eta)$, at the interior and end cluster points, a further $k+2$ linear independent equations in the β 's are obtained and hence there are now $2k+3$ linear independent equations for the $(N+1) = (2k+3)$ β 's. This system is solved for the β 's by standard Gaussian elimination techniques and hence all the constants of Eq. (22) are uniquely determined.

There are several advantages to the use of the series of complementary error functions:

- (1) $f(n)$ is positive, finite and non-zero, i.e., y will always increase with increasing n ;
- (2) $f(n)$ is continuous and successively differentiable and integrable; and
- (3) if the pivot points are spaced at a greater distance (in computational space) than $a/2$ from each other, the complementary error functions will have minimal interaction. Thus, the width and location of a complementary error function can be changed without affecting other complementary error functions.

The strategy for using this technique is as follows:

- (1) Determine the location of the interior and end cluster points in physical and computational space. The criteria used in determining these locations is usually based upon the physical processes that need to be resolved and the number of grid points needed to resolve those processes;
- (2) The pivot points, n_{pj} , and the corresponding bandwidth parameters, a are then input. Usually the interior pivot points are located on either sides of the cluster points. If it is desired to have negligible interaction between the complementary error functions, the n 's and a 's will be chosen such that no n will be located within $a/2$ grid points of each other;
- (3) Slopes are then chosen at each interior and end cluster point. Again these are usually determined from physical considerations;
- (4) A test run is then made with a stand-alone version of SUBROUTINE ONGRID. Usually the initial results will not be precisely to the liking of the user. In general this will be due to the choice of the slope constraints which can then be changed to obtain a better quality grid point distribution; and
- (5) Once this interaction process has been completed with the slope constraints (because of item 3 in the advantages of this method that process should be rapid), slight modification of the values of the n 's and a 's should result in the desired grid point distribution.

8.0 PROTOCOL FOR GRID POINT AND BOUNDARY IDENTIFICATION

Because of the complexity of the configurations considered in this study, a protocol had to be developed so that the user could easily set up the finite difference mesh required to solve the governing equations and apply the appropriate boundary conditions. The procedure utilizes the concept of a 'grid array indicator' (FORTRAN variable IFSB (I, K), where I and K correspond to the x and z directions respectively) to type the grid points in the computational domain. Referring to Fig. 4 it can be seen that the idea is to identify the grid points according to the function they serve. The convention used to identify the grid point is as follows:

- IFSB(I,K)=1 Indicates that a grid point is an interior fluid point.
- IFSB(I,K)=2 Indicates that a grid point is an interior solid point, and thus does not effect the calculation.
- IFSB(I,K)=3 Indicates that a grid point is a noncorner boundary point.
- IFSB(I,K)=4 Indicates that a grid point is an inward corner point.
- IFSB(I,K)=5 Indicates that a grid point is a re-entrant corner point.

Thus, for instance, grid points within the cross hatched areas of Fig. 4 would all have values of IFSB=2 while all the interior points would have values of IFSB=1. The boundary points (both on the cross hatched areas and on the outer boundaries) would have values of IFSB=3 except for the six inward corner points (IFSB=4), and the four re-entrant corner points (IFSB=5). The FORTRAN variable is dimensioned IFSB(71,101). Based upon the information supplied by the user, subroutine CORNER sets up the necessary parameters to control the ADI procedure. The grid array indicator is introduced through NAMELIST \$READ6.

In addition to the explanation of the grid array indicator, it is convenient at this point to explain the surface number (KSURF) convention

and direction (IDIR) convention utilized throughout the MINT computer code. The correct data input for several variables, e.g., IBOUND(KSURF, IDIR), require a knowledge of this convention. The boundary surfaces are numbered according to a surface number relative to the computational direction. The convention is to allow values of IDIR=1 and 3 to correspond to the x and z physical directions, respectively. Referring to Fig. 4, it can be seen that the surfaces (with respect to the x-direction) are numbered 1x to 10x. This corresponds to values of KSURF ranging from 1 to 10 and a value of IDIR corresponding to the x-direction, i.e., IDIR=1. The analogous convention is also applied to the z-direction surfaces. Thus, the value of KSURF represents the surface number relative to the given direction, IDIR.

9.0 MINT INPUT

Except for optional initial title cards, the entire MINT input is entered by means of the NAMELIST format. There are two primary advantages to the use of the NAMELIST format: (1) if the default values (defined by data statements) are acceptable, the user need not input that variable, and (2) the order (within a given NAMELIST) in which the input variables are entered is irrelevant. There are six NAMELIST input files in the MINT computer code, \$READ1 through \$READ6. NAMELIST \$READ1 is used to enter restart information while NAMELISTS \$READ2 through \$READ6 respectively enter geometric data, flow conditions, time step and print control, boundary conditions and initial conditions. A detailed description of all the MINT computer code input will be given below.

	Namelist or Variable Name	Description
9.1	<u>\$READ1</u>	<u>Restart Options</u>
	IREST	Restart flag. IREST=0: New calculation IREST=1: Case is being run from a restart file Default value is 0.
	IOTAPE	Output unit number for dependent variable array restart data Default value is 10.
	INTAPE	Input unit number for dependent variable array restart data Default value is 10.
	IOTAPI	Output unit number for namelist restart data Default value is 20.
	INTAPI	Input unit number for namelist restart data Default value is 20.
9.2	<u>\$READ2</u>	<u>Geometric and Grid Options</u>
	NUMDX	Number of interior grid points in the transverse direction (x direction). Total number of points in this direction = NDUMX + 2 No default value.
	NUMDZ	Number of interior grid points in the streamwise direction (z direction). Total number of points in this direction = NUMDZ + 2 No default value.
	XGMIN(IDIR)	For IDIR = 1, dimensionless value of x-coordinate on bottom boundary. For IDIR = 3, dimensionless value of z-coordinate on left boundary Default values are 3*0.0.
	XGMAX(IDIR)	For IDIR=1, dimensionless value of x-coordinate on top boundary. For IDIR=3, dimensionless value of z-coordinate on right boundary Default values are 3*1.0.

IGEOM(IDIR) Geometry option sentinel.
 IGEOM(IDIR)=1 - Grid points equally distributed in direction IDIR.

IGEOM(IDIR)=2 - Grid points distributed in direction IDIR by method of Oh.

IGEOM(IDIR)=3 - Not used.

IGEOM(IDIR)=4 - Geometric data read from file NTAG=21. If this option is used, all values of IDIR must have IGEOM(IDIR)=4.

IGEOM(IDIR)=5 - Grid points distributed on lower and upper surfaces by method of Oh. IDIR=3 only. Coordinate system will consist of straight lines drawn from corresponding points on lower and upper surface. Remainder of coordinate system will consist of lines drawn parallel to lower (or upper surface) IGEOM(1) must then equal either 1 or 2.

Default values are 3*1.

LSHAPE Shape of computational domain

LSHAPE=0 - Rectangular computational domain.

LSHAPE=2-4 - Not used.

LSHAPE=5 - Knives are input along lower surface. Location of the knife re-entrant corners are input via variables LCORNX and LCORNZ. Maximum number of corners is NCRMAX=10, i.e., five knives.

LSHAPE=6 - Computational domain is input through the use of the grid array indicator, IFSB (see NAMELIST \$READ6).

Default value is 0.

LCORNX(I) Grid point number of x location of Ith corner.

No default values.

LCORNZ(I)	Grid point number of z location of I th corner. No default values.
NCLUST(ISURF)	Number of internal cluster points for SUBROUTINE OHGRID. When ISURF=1 points are distributed in x direction. Values of ISURF=2 and 3 correspond to the distribution of grid points on the bottom and top surfaces respectively. Maximum number of internal cluster points equals 18 No default values.
CLPX(I,ISURF)	I th grid point number corresponding to the cluster points on surface ISURF. The first grid point on a surface must always be 1 while the last grid point on the transverse surface must be NDUMX + 2, and the last grid point on the streamwise surface must be NDUMZ + 2. $(I = NCLUST(ISURF) + 2) \leq 20.$ No default values.
CLFY(I,ISURF)	The x or z coordinates (corresponding to ISURF=1 and ISURF=2 and 3 respectively) which pair with the CLPX values. The first grid point on a surface must be the corresponding value of XGMIN(IDIR) and the last the corresponding XGMAX(IDIR) values. No default values.
SLOPOH(I,ISURF)	I th spacing constraint on surface ISURF. One value must be input for each CLPX(I,ISURF). However, if a negative value is input for the first and/or last values of CLPX(I,ISURF), no slope constraint will be enforced at these location(s). No default values.

ETAPOH(J,ISURF) J^{th} location of a pivot point. At the first and last values of CLPX(I,ISURF), one ETAPOH(J,ISURF) must be input. Corresponding to each interim cluster point two ETAPOH's must be input. The values of ETAPOH are the grid point location of the centers of the series of complementary of error functions. $(2*\text{CLUST}(\text{ISURF}) + 2)$ values of ETAPOH must be input for each value at ISURF. The rate of grid spacing will have the fastest variation at the pivot points. Pivot point(s) corresponding to the end points where values of SLOPOH are negative, are ignored, but must be input.

$J \leq 38$

No default values.

ALPHOH(J,ISURF) J^{th} width spacing corresponding the location of a complementary error function at ETAPOH(J,ISURF). The width spacing is the number of grid points in which 90% of the grid spacing takes place around the pivot point. Again $(2*\text{CLUST}(\text{ISURF}) + 2)$ values are required. Values of ALPHAH corresponding to the end points, where values of SLOPOH are negative, are ignored but must be input.

$J \leq 38$

No default values.

ICOORD Sentinel which determines if Cartesian or cylindrical polar formulation of governing system of partial differential equations are utilized.

ICOORD=0 - Cartesian formulation.

ICOORD=1 - Cylindrical polar formulation.

Default value is 0.

INOUT In core - Out of core option.
INOUT=0 - In core.
INOUT=2 - Out of core option.

Default value is 2.

9.3 SREAD3

IUNITS System of units utilized.
IUNITS=1 - MKS.
IUNITS=2 - English.
Default value is 1.

CLENG	Reference length (m or ft).
	No default.
MINF	Free stream Mach number.
	No default.
TINF	Free stream static temperature (°K or °R). Input required if REPL=0.0.
	No default.
PINF	Free stream static pressure (Nt/m^2 or lbf/ft^2). Input required if REPL=0.0. If REPL≠0.0, either PINF or PZERO must be input.
	No default.
REPL	Reynolds number per unit length (m^{-1} or ft^{-1}).
	No default.
PZERO	Free stream stagnation pressure (Nt/m^2 or lbf/ft^2).
	No default.
INSTAG	Enthalpy option. INSTAG=0 - Energy equation is formulated in terms of a static enthalpy.
	INSTAG=2 - Constant stagnation enthalpy is assumed.
	Default value is 0.
ISWIRL	Swirl equation option. ISWIRL=0 - No swirl equation is solved.
	ISWIRL=1 - Swirl equation is solved - used for cases with rotation. ICOORD must equal 1.
	Default value is 0.
IVISC	Viscosity model option. IVISC=1 - Constant viscosity.
	IVISC=2 - Laminar viscosity - Sutherland's law.
	IVISC=3 - Mixing length turbulent viscosity with Sutherland's laminar viscosity.
	IVISC=4 - $k-l_m$ turbulent viscosity with Sutherland's laminar viscosity.

IVISC=5 - $K-\epsilon$ (Jones-Launder) turbulent viscosity with Sutherland's laminar viscosity

IVISC=-5 - Solve $K-\epsilon$ equations only with frozen fluid dynamics.

Default value is 1.

FK Wall roughness factor.

Default value is 0.0.

AVISC(IDIR,IEQ) Artificial dissipation parameter I (See Eq. (1) of Appendix II) IEQ=1-5, 16 and 17 corresponding to the x,y,z momentum, continuity energy, k and ϵ equations respectively

Default values are all 0.5.

9.4 \$READ4

Time Step and Print Control

NT Number of time steps to be run.

Default value is 0.

DTMIN Minimum nondimensional time step for this run.

No default value.

DTMAX Maximum nondimensional time step for this run.

No default value.

DT Initial nondimensional time step. If DT is omitted on a restart, DT will be set to the value at termination of last run.

No default value.

IDTADJ

Time step control.

IDTADJ=0 - Constant DT is used for this run.

IDTADJ=1 - Time step adjusted. If maximum relative change in any flow variable is less than 0.04, DT is multiplied by 1.25 to a maximum of DTMAX. If maximum relative change in any flow variable is greater than 0.06, DT is divided by 1.25 to a minimum of DTMIN.

IDTADJ=2 - Time step is cycled between DTMIN and DTMAX over NTSTEP time steps.

Default value is 0.

NTSTEP	Number of time steps used in cycling. Default value is 5.																																
NTREST	Restart is written every NTREST time steps. No default value.																																
IPRINT	Complete flow field printout is provided every IPRINT time steps. No default values.																																
IVARPR(IV)	Print control flag for variable IV. IVARPR(IV)=0 - Suppresses printout of variable. IVARPR(IV)=1 - Printout variable IV occurs. <table> <thead> <tr> <th><u>IV</u></th> <th><u>IVARPR(IV)</u></th> </tr> </thead> <tbody> <tr> <td>1</td> <td>Transverse velocity - U</td> </tr> <tr> <td>2</td> <td>Swirl velocity - V</td> </tr> <tr> <td>3</td> <td>Streamwise velocity - W</td> </tr> <tr> <td>4</td> <td>Density - ρ</td> </tr> <tr> <td>5</td> <td>Enthalpy - h or h_0</td> </tr> <tr> <td>16</td> <td>Turbulence kinetic energy - k</td> </tr> <tr> <td>17</td> <td>Dissipation of turbulence - ϵ</td> </tr> <tr> <td>26</td> <td>Static pressure - P</td> </tr> <tr> <td>27</td> <td>Static temperature - T</td> </tr> <tr> <td>28</td> <td>Effective viscosity - μ_{eff}</td> </tr> <tr> <td>29</td> <td>Total temperature - T_0</td> </tr> <tr> <td>33</td> <td>Mixing length - L_n</td> </tr> <tr> <td>34</td> <td>Dissipation - D:D</td> </tr> <tr> <td>35</td> <td>Mach number - M</td> </tr> <tr> <td>36</td> <td>Stagnation pressure - P_0</td> </tr> </tbody> </table>	<u>IV</u>	<u>IVARPR(IV)</u>	1	Transverse velocity - U	2	Swirl velocity - V	3	Streamwise velocity - W	4	Density - ρ	5	Enthalpy - h or h_0	16	Turbulence kinetic energy - k	17	Dissipation of turbulence - ϵ	26	Static pressure - P	27	Static temperature - T	28	Effective viscosity - μ_{eff}	29	Total temperature - T_0	33	Mixing length - L_n	34	Dissipation - D:D	35	Mach number - M	36	Stagnation pressure - P_0
<u>IV</u>	<u>IVARPR(IV)</u>																																
1	Transverse velocity - U																																
2	Swirl velocity - V																																
3	Streamwise velocity - W																																
4	Density - ρ																																
5	Enthalpy - h or h_0																																
16	Turbulence kinetic energy - k																																
17	Dissipation of turbulence - ϵ																																
26	Static pressure - P																																
27	Static temperature - T																																
28	Effective viscosity - μ_{eff}																																
29	Total temperature - T_0																																
33	Mixing length - L_n																																
34	Dissipation - D:D																																
35	Mach number - M																																
36	Stagnation pressure - P_0																																

Default values are 1, 0, 1, 12*0, 2*1, 8*0, 3*1, 4*0, 1, 0, 2*1.

IGPRT(IV)

Geometric variable printout option.

IGPRT(IV)=0 - Suppresses printout of variable IV.

IGPRT(IV)=1 - Printout of variable IV occurs on initial run.

IGPRT(IV)=2 - Printout of variable IV occurs before first step of each run.

<u>IV</u>	<u>IGPRT(IV)</u>
-----------	------------------

1	$J \frac{\partial y^1}{\partial x_1}$
3	$J \frac{\partial y^3}{\partial x_1}$
7	$J \frac{\partial y^1}{\partial x_3}$
9	$J \frac{\partial y^3}{\partial x_3}$
13	r
17	J
21	X
23	Z

Default values are 12*0, 3*1,
0, 3*1, 0, 6*1.

IPLOT

Plot option.

IPLOT=0 - No plot file written.

IPLOT=999 - Plot file written on file TAPE1.

Default value is 0.

ATIME1, ATIME2, ZD1,
ZD2, ZD3, ZD4Input for spatial variation of time step - z
direction only.

See Eq. (14)

Default values are:

ATIME1=0.0,	ATIME2=0.0
ZD1=10000.0	ZD2=-10000.0
ZD3=10000.0	ZD4=-10000.0

ISETKE

Time step when k-e equations are first solved
No default value.

IBOUND(ISURF, IDIR)Boundary Condition Input

Type of boundary on surface number ISURF corresponding to direction IDIR.

IBOUND(ISURF, IDIR)=1 - Inlet

2 - Symmetry

3 - Exit

4 - Wall

Maximum number of values in each IDIR direction is 50.

No default values.

IEQBC(ISURF, IDIR, IEQ)

Boundary condition on surface ISURF corresponding to direction IDIR associated with equation IEQ. IEQ has values of 1,2,3,4,5,16 and 17 corresponding to x,y,z momentum equations, continuity, energy, k and ϵ equations respectively. The defaulted values of IEQBC are determined by the values of IBOUND(ISURF, IDIR).

For IBOUND (ISURF, IDIR)=1 the cross flow velocity components are assumed to remain unchanged. The streamwise velocity component is determined from the two layer model. The pressure is extrapolated and the stagnation enthalpy remains constant. Values of k and ϵ also remain unchanged. For IBOUND(ISURF, IDIR)=2 symmetry conditions are used, i.e., the first derivative of all variables except the normal velocity component are zero. The normal velocity component is zero.

For IBOUND(ISURF, IDIR)=3 exit conditions are assumed. The second derivative of all velocity components, temperature, k and ϵ are set to zero. The static pressure is prescribed.

For IBOUND(ISURF, IDIR)=4 the no slip conditions are set for the velocity components. The pressure condition is set by assuming that the first derivative of that variable is zero. The adiabatic condition is assumed for the energy equation unless a nonnegative value of TWALL(ISURF, IDIR) is input, in which case the wall temperature is set to that value. Any default value IEQBC,(ISURF, IDIR, IEQ), can be overwritten by inputting a nonzero value of IEQBC.

<u>IEQBC (ISURF, IDIR, IEQ)</u>	<u>Boundary Conditions</u>
1	No change in function.
2	First derivative equals zero.
3	Second derivative equals zero.
4	Rough surface.
19	Two-layer model momentum equation.
50	Normal injection velocity specified.
51	Normal mass flux specified.
52	Rotational velocity specified.
53	Rotational profile on inlet.
LZINJ1	z-direction grid point where injection begins. No default.
LZINJ2	z-direction grid point where injection ends. No default.
NTINJ1	Time step when injection begins. No default.
NTINJ2	Time step when injection reaches final value. No default.
VXINJ	Injection velocity (IEQBC=50) or injection mass flux (IEQBC=51). No default.
NTROT1	Time step when rotation begins. No default.
NTROT2	Time step when rotation reaches final value. No default.

VROT	Rotational speed (radians/sec).
	No default.
NPRES1	Time step when back pressure begins to change.
	Default value is 0.
NPREP2	Time step when back pressure reaches final value.
	No default.
PAMB	Ambient pressure.
	No default.
TWALL(ISURF, IDIR)	Dimensional wall temperature, °K or °R.
	Default values are 150*0.0.
CF1	Skin friction coefficient on bottom surface of inlet. Needed for turbulent flow.
	No default.
CF2	Skin friction coefficient on top surface of inlet. Needed for turbulent flow.
	No default.
DELTA1	Boundary layer thickness on bottom surface of inlet.
	No default.
DELTA2	Boundary layer thickness on top surface of inlet.
	No default.
TEMPW1	Temperature of bottom surface, °K or °R.
	No default.
TEMPW2	Temperature of top surface, °K or °R.
	No default.
DELTR	Thickness of rotational boundary layer on rotor. The rotational velocity (nondimensional) in the inlet will be a maximum of VROT*XGMIN(1) on the rotor and decrease by means of a cosine function to zero at XGMIN(1) + DELTR.
	No default value.

9.6 \$READ6

IFSB(I,K)

Grid Indicator

Indicator to type the grid points in the computational domain. I corresponds to the $x(y^1)$ direction and K corresponds to the $z(y^3)$ direction. Only used when LSHAPE=6.

IFSB(I,K)=1 - Interior fluids point.

IFSB(I,K)=2 - Interior body point.

IFSB(I,K)=3 - Boundary point.

IFSB(I,K)=4 - Boundary point - re-entrant corner.

IFSB(I,K)=5 - Boundary point - inward corner.

Default values are 7171*2.

10.0 SAMPLE INPUT

Sample input for the MINT computer code is presented in Table 2. The case considered is that of a worn single knife seal (case 6a of Ref. 1). The first card of the input is an identifier for the input. The remainder of the input is entered through NAMELISTS \$READ1 through \$READ6. This particular run is for a restart (IREST=1). However, except for the IREST=1 card, the input for the initial run would be identical. NAMELIST \$READ2 indicates that the geometry utilized in this run is obtained from file TAPE21 (IGEOM=3*4) and that a two-dimensional coordinate system is used (ICOOND=0). Values of NUMDX=29 and NUMDZ=99 indicate that 31 and 101 grid points are used respectively in the x and z directions. The nondimensional dimensions of the physical domain are given by XGMIN and XGMAX as 12.0 and 300.0 in the x and z direction, respectively. A value of LSHAPE=0 indicates that no reentrant corners occur in the domain. NAMELIST \$READ3 gives the reference conditions through FORTRAN variables CLENG, MINF, TINF and REPL in English units (IUNITS=1). A value of IHSTAG=0 indicates that the energy equation is to be solved while a value of IVISC=5 indicates that a k- ϵ turbulence model is used. NAMELIST \$READ4 indicates that this case is to be run for 50 time steps (NT=50) from the last restart. This run is to be made at a constant time step of DT=0.05, DTMIN=0.05, DTMAX=0.05 and IDTADJ=0. Printout will occur each 25 time steps (IPRINT=25), and variables u, w, k, ϵ , p, T, u_{eff} and M will be printed. ISETKE=650 indicates that this is the time step when the k- ϵ equations were first solved. NAMELIST \$READ5 indicates that the upper and lower surfaces are walls and that the left hand side is an inlet and the right hand side is an exit (IBOUND=2*4, 48*0, 50*0, 1,3,48*0). The default boundary conditions are used. Values of PAMB=0.5, NPRES1=0 and NPRES2=50 indicate that the back pressure was dropped from the total source pressure to a value equal to 0.5 of the reference pressure over the first 50 steps of the run. CFI=CP2=5.0E-03, DELTA1=DELTA2=0.5 and TEMPW1=TEMPW2 are the parameters used for the boundary layer profile on the inlet surfaces (rotor and land, respectively). NAMELIST \$READ6 is empty as the LSHAPE=0 is used. However, the \$READ6 and \$END cards must still be input.

11.0 SAMPLE OUTPUT

Sample output is presented in Table 3. The output consists of (1) the time step history of the run, and (2) the resulting output. Only sample portions of the output are presented as additional output would be repetitive and serve no purpose.

The time step history output consists of pertinent output needed to monitor the run to determine the performance (convergence) of the run. It consists of the time step information (time step number, total CP time for this run, the physical elapsed time, the time step, the viscous stability criterion and the value of IDTADJ). The next two lines contain the maximum normalized change during a time step (SSTST) and its grid point location. The remainder is the maximum normalized change for all variables. The last portion of the time step history output contains the residual information. This residual information consists of the maximum average and peak residual for all equations and of the average and maximum residual, with its grid point location, for each equation.

The second portion of the output consists of the various output as determined by the FORTRAN input variable IVARPR. For economy only the streamwise velocity, W-VEL, and the turbulence kinetic energy, TKE, are presented.

12.0 PLOT PROGRAM USER'S MANUAL

The MINT computer program does not have a plotting capability. Rather a plot file is written in SUBROUTINE WRPLOT and the user is free to interface with any desired plotting program. A brief description and user's manual for a plotting program, TDPLOTS is presented below. Program TDPLOTS was specifically developed to interface with the MINT computer code, hence the plot file written in SUBROUTINE WRPLOT directly interfaces with TDPLOTS. This code has the capability of producing plots of (1) the coordinate system, (2) vector plots of the velocity components, (3) contour plots of up to 10 variables and (4) profile plots of up to 10 variables. TDPLOTS has a general three-dimensional capability, however in this description only the two dimensional option will be discussed. All plotting in program TDPLOTS is done in an x-y coordinate system where, by convention, x is the abscissa and y is the ordinate. In the discussion of the input variables, that convention will be maintained. This program has the capability of zooming in on any rectangular area of the physical domain. The input of FORTRAN variable control the zoom capability so that it is not hardware limited. The subroutines of program TDPLOTS are written so that they are not limited to a particular graphics library. This is done by having all interface logic pass through one subroutine, PLTPKG, which must be specialized for each library. At present four versions of subroutine PLTPKG exist and can be used on the following graphics library systems: (1) CALCOMP, (2) IBM 3033 - NASA Lewis, (3) PLOTPSE and (4) META. The input for program TDPLOTS is input by means of the NAMELIST format. Three NAMELISTS \$PLOT, \$PLOT2 and \$PRPPLT are utilized. A description of the input associated with these NAMELIST's will be given below. This will be followed by sample input.

<u>Namelist on Variable None</u>	<u>Description</u>
12.1 <u>\$PLOT</u>	<u>Control Input</u>
NCASE	Number of cases to be run. The number of cases is defined as the number of zooms required. In each case information must be input with NAMELIST \$PLOT2 Default value is 1.

12.2 \$PLOT2Case Options

ICOORD	Coordinate plot option. ICOORD=0 - No plot. ICOORD=1 - Plot. Default value is 1.
IVECT	Velocity vector plot option. IVECT=0 - No plot. IVECT=1 - Plot. Default value is 0.
VECM	Scaling for velocity vectors; length of tail of velocity vectors will be scaled by VECM. If VECM=0.0, scaling will be done on maximum value of magnitude of the velocity in the computational domain. Default value is 0.0.
IKILLX(I)	Parameter to cull specific velocity vectors in the transverse direction at Ith grid point. IKILLX(I)=0 - Cull IKILLX(I)=1 - No cull. Default values 101*1.
IKILLY(J)	Parameter to cull specific velocity vectors in the normally streamwise direction at the Jth grid point. IKILLY(K)=0 - Cull. IKILLY(K)=1 - No cull. Default values are 101*1.
IPLOT(IV)	Contour plot option IPLOT(IV)=0 - No plot. IPLOT(IV)=1 - Plot. Default values are 10*0.

<u>IV</u>	<u>Contour Parameter</u>
1	U
2	V
3	W
4	ρ
5	P
6	T
7	M
8	ψ (stream function)
9	C_p
10	C_{p_0}

NHTS(IV) Number of contour levels for variable IV.
Maximum 10.

Default values are 10*10.

IHO(IV) Sentinel to determine if contour levels are to be input or calculated.

IHO(IV)=0 - Contour levels calculated. Maximum and minimum levels of the contour variable are determined by scanning the computational domain. Plotted contour levels will then be equally spaced between the maximum and minimum at NHTS(IV) - levels.

IHO(IV)=1 - Contour levels input from variable HTS(I,IV).

Default values are 10*0.

HTS(I,IV) Contour levels; value of I goes from 1-NHTS(IV).

Default values are 100*0.0.

ISCL Zoom parameter.

ISCL=0 - No zoom.

ISCL=1 - Zoom.

Default value is 0.

XSCL1	First x coordinate location for zoom (used only if ISCL=1). If XSCL1 is positive, plot (coordinate, velocity or contour plot(s)) start at XMN+XSCL1 on the x axis. Here XMN is the minimum value of x in the field. If XSCL1 is negative, plot will start at XMX-XSCL1 where XMX is the maximum value of x in the field.
	Default value is -1×10^{50} .
XSCL2	Last x coordinate location for zoom (used only if ISCL=1). If XSCL2 is positive, last x coordinate is XMN+XSCL2. If XSCL2 is negative, last x coordinate is XMX-XSCL2.
	Default value is 1×10^{50} .
YSCL1	First y coordinate location for zoom; analogous to XSCL1.
	Default value is -1×10^{50} .
YSCL2	Last y coordinate location for zoom; analogous to XSCL1.
	Default value is 1×10^{50} .
IXB1	Sentinel to draw lower boundary of plot domain.
	IXB1=0 - No draw.
	IXB1=1 - Draw.
	Default value is 1.
IXB2	Sentinel to draw top boundary of plot domain.
	IXB2=0 - No draw.
	IXB2=1 - Draw.
	Default value is 1.
IYB1	Sentinel to draw left hand boundary of plot domain.
	IYB1=0 - No draw.
	IYB1=1 - Draw.
	Default value is 1.

	IYB2	Sentinel to draw right hand boundary of plot domain. IYB2=0 - No draw. IYB2=1 - Draw. Default value is 1.
	NPRFIG	Number of profile plots. If NPRFIG=0, NAMELIST PRFPLT input is omitted. Maximum number of figures is 10. Default value is 0.
12.3	<u>\$PRFPLT</u>	<u>Profile Plot Input</u>
	NPRF(IP)	Number of profile plots on IP th figure. Default values are 10*1.
	NVRPL(IV,IP)	Variables to be plotted on IP th profile plot. Default values are 100*0.
		<u>Variable</u> <u>NVRPL(IV,IP)</u>
		U 1
		V 2
		W 3
		P 4
		P 5
		T 6
		M 7
		ψ 8
		C _P 9
		C _{P0} 10
	CRDI(I,IP)	CRDI(1,IP) - x coordinate of the first profile point of the IP th plot. CRDI(2,IP) - x coordinate of the last profile point of the IP th plot. No default values.

CRD2(I,IP)	CRD2(1,IP) - y coordinate of the first profile point of the IP th plot.
	CRD2(2,IP) - y coordinate of the last profile point of the IP th plot.
	No default values.
NPTS	Number of equally spaced points used on straight line joining first and last profile points to obtain the interpolated values for the profile plot(s). Maximum value of 101.
	Default value is 50.
XOFSET(IP)	Constant to be added to profile plot independent variable. If XOFSET=0 independent variable will vary from 0 to the length of the line joining the first profile point (CRD1(1,IP), CRD2(1,IP)) to the last profile point (CRD(2,IP), CRD2(2,IP)).
	Default values are 10*0.0.
DSCALE(IP)	Scale factor used to multiply dependent variables for IP th plot. Often used to plot dimensional values.
	Default values are 10*1.0.
XLEN(IP)	Length (in) of x axis.
	Default value is 10*6.0.
YLEN(IP)	Length (in) of y axis.
	Default value is 10*6.0.
ITURN(IP)	Flag used to rotate IP th profile plot.
	ITURN(IP)=0 - Dependent variable along x-axis.
	ITURN(IP)=1 - Dependent variable along y-axis.
	Default values are 10*0.

PIMIN(IP), PIMAX(IP)	Extents of independent variable axis for IP th plot. Note that the inch increment on the x-axis will be equal to (PIMAX(IP) - PIMIN(IP))/XLEN(IP) of independent variable.
PDMIN(IP), PDMAX(IP)	Extents of dependent variable axis for IP th plot. Note that the inch increment on the y-axis will be equal to (PDMAX(IP)- PDMIN(IP))/YLEN(IP) of independent variable.

Default is automatically calculated using the dependent variable values to be plotted.

12.4 Sample Input

Sample TDPLOTS input is presented in Table 4. The NAMELIST \$PLOT input NCASE=2 denotes that two cases are to be run thus requiring that NAMELIST \$PLOT2 must be input twice. The first NAMELIST \$PLOT2 uses the zoom option (ISCL=1), and will make plots between x-values of 12.0 to 18.0 and y-values of 0.0 to 10.0. Since the default value of ICOORD is 1, a coordinate plot will be generated. The values of IPLOT will cause standard contours (10 contour levels between the minimum and maximum field values) to be plotted for Mach number and stream function. No vector or profile plots are associated with the first \$PLOT input, i.e., default values of IVECT and NPRFIG are both zero. The second NAMELIST \$IPLOT2 input will retain the previously defined ISCL=1 value, and will plot contours of Mach number and stream function between x-values of 18.0 to 24.0 and y-values of 0.0 to 10.0 (the values from the first \$PLOT2 input are retained). In addition, vector plots will be made over the same domain (IVECT=1). However, the coordinate plot will not be made (ICOORD=0). Two profile plots will be made (NPRFIG=2) necessitating \$PRFPLT input. Both plots will contain two profiles (NPRF(1)=2*2) of u and w velocity components (NVRPL(1,1)=1,3,8*0,1,3,8*0) at a constant x station of 20.0 with y-values varying between 0.0 and 10.0.

REFERENCES

1. Buggeln, R.C. and McDonald, H.: Labyrinth Seal Analysis, Volume I - Development of a Navier-Stokes Analysis for Labyrinth Seals, AFWAL-TR-85-2103, Vol. 1, Scientific Research Associates, Inc., Glastonbury, CT, May 1985.
2. Oh, Y.N.: An Analytic Transformation Technique for Generating Uniformly Spaced Computational Mesh, Final Report, NASA-Langley Research Grant NSG1087, October 1978.
3. Schlichting, H.: Boundary-Layer Theory, Sixth Edition, McGraw-Hill Book Company, 1968, pp. 192-193.
4. Maise, G. and McDonald, H.: Mixing Length and Kinematic Eddy Viscosity in a Compressible Boundary Layer. AIAA Journal, Vol. 6, No. 1, January 1968, pp. 73-80.

LIST OF SYMBOLS

Symbols

B	Constant from quadratic equation (Eq. 9)
b	Blockage factor
c_p	Specific heat at constant pressure
c_1	Constant in Sutherland's viscosity law
c_2	Constant in Sutherland's viscosity law
f	Cosine factor or transformation function
h	Enthalpy
l	Length
M	Mach number
n	Time step number
P	Pressure
R	Universal gas constant
Re	Reynolds number
r	Radius
T	Temperature
t	Time
U	Velocity
V_θ	Rotational velocity
y	Physical distance
z	Streamwise distance

Greek Symbols

α	Computational width factor
β	Step height for pivot
γ	Ratio of specific heats or scaling constant

Δ Change
 η Computational distance
 θ Grouping defined Eq. (23)
 μ Viscosity
 ρ Density

Subscripts

a Ambient
B Back or knife assembly
f Final
jth
o Stagnation; first
p Pivot
1 First
2 Second
3 Third
4 Fourth
max Maximum
min Minimum
∞ Free stream or reference condition

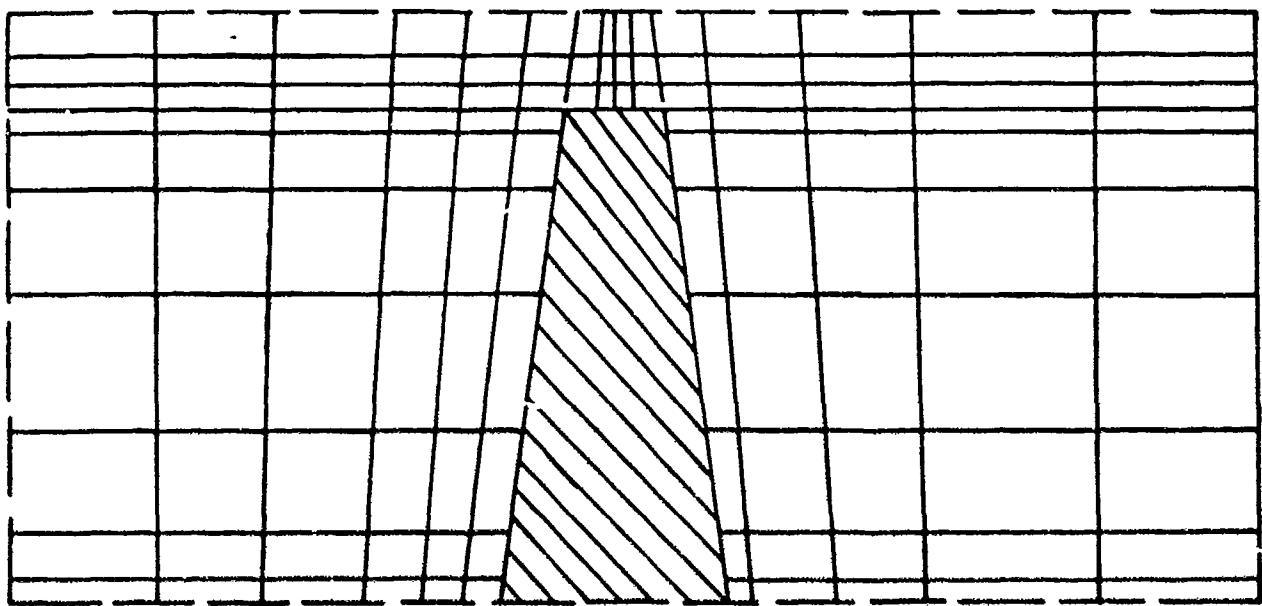


Figure 1 - Typical Coordinate System Construction.

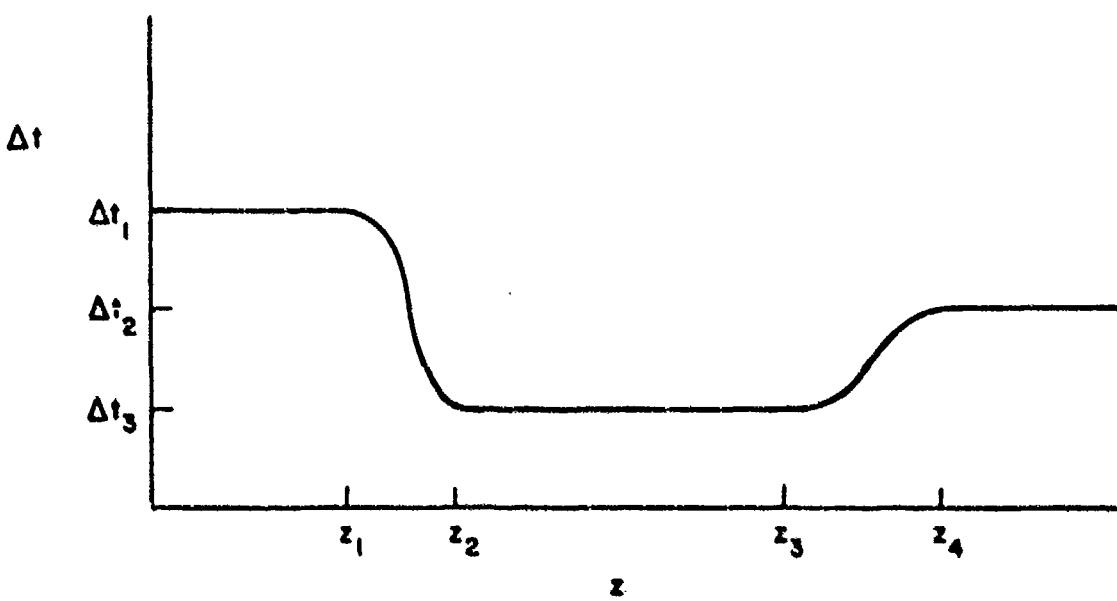


Figure 2 - Time Step Selection.

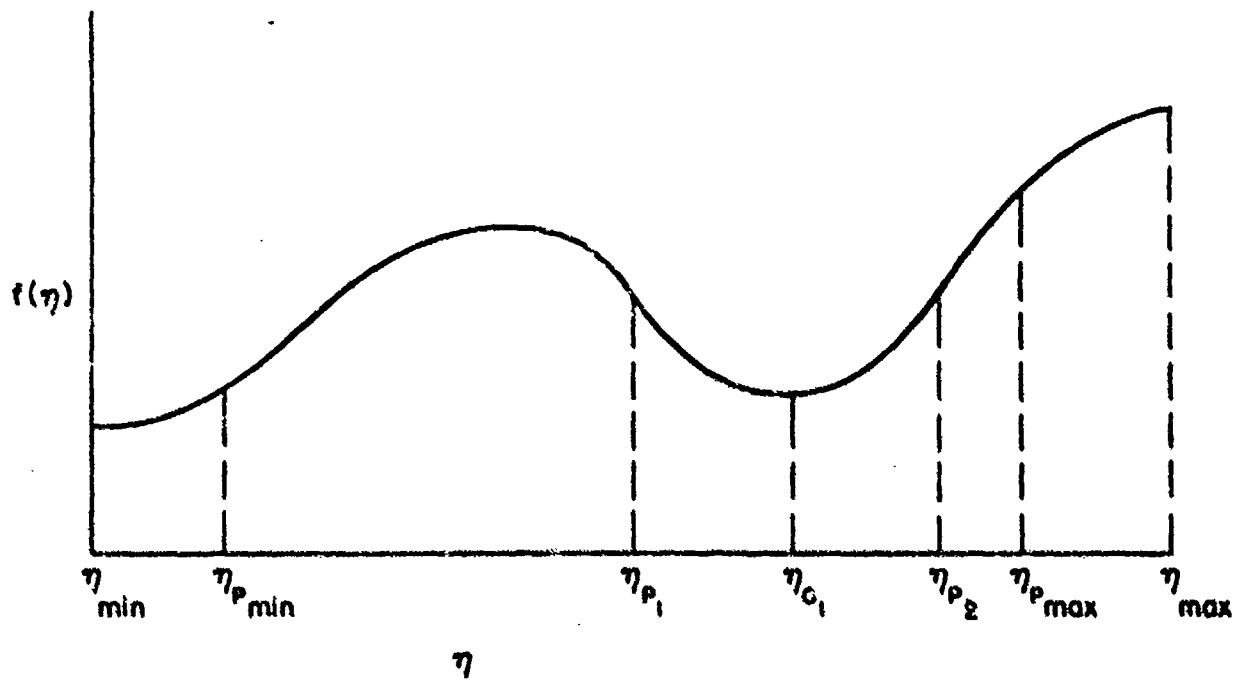


Figure 3 - Oh Grid Notation.

○ - INWARD CORNER POINT
● - RE-ENTRANT CORNER POINT

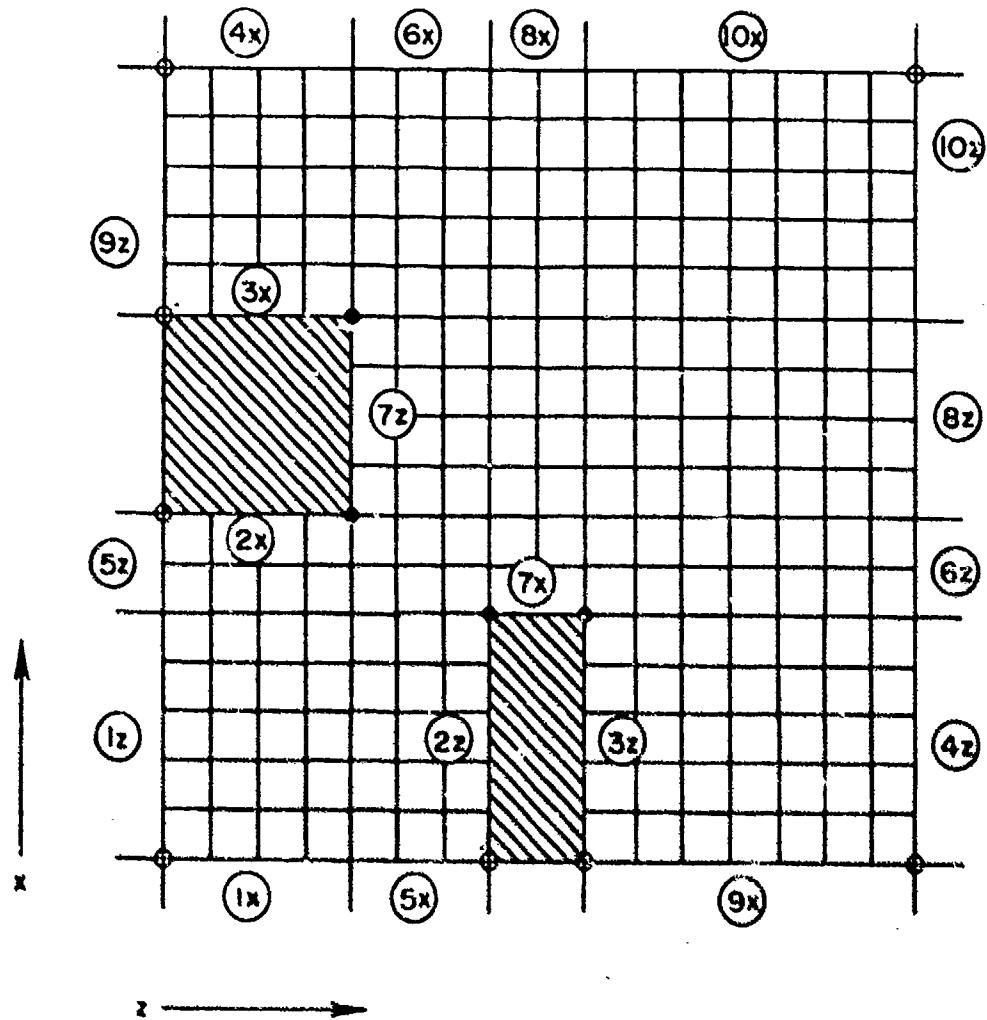


Figure 4 - A Sample Computational Domain With Embedded Solid Bodies.

Table 1 - Reference Variables - Units

<u>UNITS</u>			
<u>VARIABLE</u>	<u>UNITS</u>	<u>MKS</u>	<u>ENGLISH</u>
l_∞	l	m	ft
ρ_∞	$\frac{\text{mass}}{l^3}$	$\frac{\text{kg}}{m^3}$	$\frac{\text{slugs}}{ft^3}$
T_∞	Deg	$^{\circ}\text{K}$	$^{\circ}\text{R}$
P_∞	$\frac{f}{l^2}$	$\frac{nt}{m^2}$	$\frac{lbf}{ft^2}$
U_∞	$\frac{l}{t}$	$\frac{m}{sec}$	$\frac{ft}{sec}$
R	$\frac{l-f}{\text{mass-Deg}}$	$2.871 \times 10^2 \frac{nt \cdot m}{kg \cdot K}$	$1.715 \times 10^3 \frac{ft-lbf}{slug \cdot ^{\circ}\text{R}}$
μ_∞	$\frac{\text{mass}}{l-t}$	$\frac{kg}{m \cdot sec}$	$\frac{\text{slugs}}{ft \cdot sec}$
h_∞	$\frac{l^2}{t^2}$	$\frac{m^2}{sec^2}$	$\frac{ft^2}{sec^2}$
c_{p_∞}	$\frac{l^2}{t^2 \text{Deg}}$	$\frac{m^2}{sec^2 \cdot K}$	$\frac{ft^2}{sec^2 \cdot ^{\circ}\text{R}}$

Table 2 - Sample MINT Input

```
****WORN SINGLE KNIFE SEAL****
$READ1
IREST = 1,
$END
$READ2
NUMDX = 29, NUMDZ = 99,
IGEOM = 3*4, ICOORD = 0,
XGMIN = 0.0, 0.0, 0.0,
XGMAX = 12.0, 3.0, 300.0,
LSHAPE = 0,
$END
$READ3
IUNITS = 1,
CLENG = 2.54E-03,
REPL = 2.4309269E+06,
MINF = 5.4357E-02,
PZERO = 2.004087E+05,
IHSTAG = 0,
IVISG = 5,
$END
$READ4
NT = 50,
IPRINT = 25,
IDTADJ = 1, DT = 0.05, DTMIN = 0.05, DTMAX = 0.05,
IVARPR = 1,0,1,12*0,2*1,8*0,3*1,6*0,2*1,
PAMB = 0.5, NPRES1 = 0, NPRES2 = 50,
ISETKE = 650,
$END
$READ5
IBOUND = 2*4,48*0, 50*0, 1,3,48*0,
DELTAL = 0.5, DELTA2 = 0.5, CF1 = 5.0E-03, CF2 = 5.0E-03,
TEMPW1 = 294.44, TEMPW2 = 294.44,
$END
$READ6
$END
```

Table 3 - Sample MINT Output

Table 3 - Sample MINT Output (Continued)

Table 3 – Sample MINT Output (Continued)

Table 3 - Sample MINI Output (Continued)

Table 3 - Sample MINT Output (Continued)

Table 4 - Sample Plot Input

```
$PLOT
NCASE = 2,
$END

$PLOT2
ISCL = 1,
XSCL1 = 12.0, XSCL2 = 18.0,
YSCL1 = 0.0, YSCL2 = 10.0,
IPLOT(1) = 6*0,2*1,2*0,
$END

$PLOT2
XSCL1 = 18.0, XSCL2 = 24.0,
ICOORD = 0,
IVECT = 1,
NPRFIG = 2,
$END

$PRFPLT
NPRF(1) = 2*2,
NVRPL(1,1) = 1,3,8*0,1,3,8*0,
CRD1(1,1) = 4*20.0,
CRD2(1,1) = 0.0,10.0,0.0,10.0,
$END
```